Sensors and Actuators A 199 (2013) 9–17

Contents lists available at SciVerse ScienceDirect

Sensors and Actuators A: Physical

journal homepage: www.elsevier.com/locate/sna

Experimental and numerical design of a long-waist cone flow meter

Chao Tan, Hao Wu, Can Wei, Feng Dong*

Tianjin Key Laboratory of Process Measurement and Control, School of Electrical Engineering and Automation, Tianjin University, Tianjin 300072, China

ARTICLE INFO

Article history:
Received 14 January 2013
Received in revised form 19 April 2013
Accepted 30 April 2013
Available online 15 May 2013

Keywords:
Long waist cone flow meter
CFD simulation
Differential pressure
Discharge coefficient
Flow measurement
Multiphase flow

ABSTRACT

A long-waist cone flow meter is designed for steady differential pressure measurement and relatively small overall pressure drop. A constant-diameter annular flow channel is formed between cone element and pipe wall. The flow field of water flows through cone element is studied with 3-D CFD simulations. The constant-diameter annular channel is found being able to adjust the incoming fluid, and small rear angle of cone element could reduce flow separation and overall pressure difference. An optimized structure of long-waist cone flow meters, referred as LWC, is proposed based on the investigations on flow field analysis. A steady contracting differential pressure and an overall differential pressure can be tapped from a LWC. A set of LWCs was fabricated with diameter ratio β ranging from 0.55 to 0.75, and waist length L from 20 to 60 mm when $\beta = 0.65$ for experimental tests and CFD verifications. A decreasing trend of dimensionless differential pressure Δp^* and δp^* and an increasing trend of discharge coefficient C_d of LWCs under different Reynolds number are observed and analyzed, and flow rate prediction of different Newtonian single phase flow is experimentally studied. A case study of oil–water two-phase flow is then presented with a selected LWC and an average error below 5% is achieved when treating the mixture a homogeneous flow.

1. Introduction

Flow rate of a single phase or multiphase flow is a very important process parameter in industrial processes. This topic has been investigated for decades and quite a lot of measuring techniques and flow meters were developed [1], among which the differential pressure (DP) flow meters were of wide interests by researchers and engineers, for instance the orifice plate, Venturi, nozzle and cone meters [2,3]. DP flow meters are also well studied to measure the flow rate of multiphase flow that frequently encountered in power plants, chemical engineering, petroleum production and transportation. The accurate measurement of multiphase flow rate presents a complicated problem, and the requirement for this type of measurement is still rising in both the engineering and academic world [4].

Problems associated with DP flow meters that limit their applications include unstable discharge coefficients, non-linearity, low repeatability and accuracy, high pressure loss and long straight pipes required in front of the meter [5]. Therefore optimization and re-design of the throttling set led to the emergence of non-standard DP flow meters such as wedge meters and eccentric orifice plates for dirty fluids, and hemicycle orifice plates for low Reynolds number flow [6,7]. In the 1950s the annular orifice plates were introduced which are constructed by fixing a circular plate concentrically in the pipeline forcing the fluid to flow within an annular channel, where the dense phase usually tends to flow at the bottom of a horizontal pipeline and the light phase flows at the top [8]. This configuration cannot adjust the flow condition and thus is incapable of stabilizing the flow process, which is a defect similar to that of orifice plates. In light of the advantages of the annular orifice plates, came up a cone flow meter, which, instead of contracting the flow to the central line of the pipe, pushes the flow to the pipe wall by inserting a concentric cone element inside the pipe [9]. The cone flow meters are capable of measuring the flow rate of clean or dirty fluids, wet gas [10], with a shorter straight pipeline required in front, and wider turn down ratio compared to a standard DP flow meter, but also with a relatively low accuracy and high pressure loss due to the flow wake induced at the cone tail [11]. Different structures of cone flow meters were developed, including Spindle Flow meter [12], Shuttle Type flow meter [13], to improve measurement accuracy and reduce pressure loss. They all adopt the idea of using annular flow channel for flow adjustment, and their unique features include flow adjustment, steady pressure profile, relatively low pressure loss and abrasion free.

In this work, a long-waist cone flow meter is designed for multiphase flow measurements. The design principle is to take advantage of the annular flow channel to flatten the flow pressure/velocity profile and consequently reduce the overall pressure difference and improve measurement repeatability. Furthermore, the annular channel provides an extra option for implementing a secondary

* Corresponding author.
E-mail address: fdong@tju.edu.cn (F. Dong).

0924-4247/$ – see front matter
http://dx.doi.org/10.1016/j.sna.2013.04.039

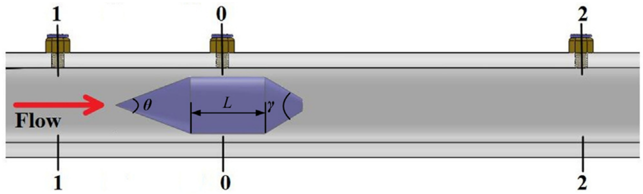

Fig. 1. Basic structure of the long-waist cone meter.

instrument for measuring phase fraction of multiphase flow thanks to the mixing effect of the cone element. As a fundamental work, the key parameters of the cone element are studied in computational fluid dynamic (CFD) simulations on their effect on flow adjustment and measurement. Structure of the long-waist cone flow meter is optimized by considering the influence of constant-diameter length, front/rear cone angle, and diameter ratio. A set of LWCs was fabricated with diameter ratio β ranging from 0.55 to 0.75, and waist length L from 20 to 60 mm when $\beta = 0.65$ for experimental tests and CFD verifications. Mineral oil is used in dynamic experiments to reduce the Reynolds number in order to evaluate the performance of LWCs in different Newtonian flows. A case study of oil–water two-phase flow is then presented with a selected LWC to further investigate their performance in two-phase flow.

2. Flow field study with CFD simulations

2.1. Basic structure and theoretical correlations of a long-waist cone flow meter

The basic structure of a long-waist cone flow meter is shown in Fig. 1, the cylinder part forms a constant-diameter annular flow channel with pipe wall and thereby inherits the advantages of annular flow meters.

A contracting pressure drop Δp (between pressure tap 1 and 0) is produced as fluid flows through the head of cone element into the constant-diameter annular channel, Δp is related to mass flow rate W by Bernoulli's equation and continuity equation:

$$W = \frac{\xi C_d}{\sqrt{1-\beta^4}} A_0 \sqrt{2\Delta p \rho} \quad (1)$$

where ξ is compressibility, $\beta = \sqrt{D^2 - d^2}/D$ is equivalent diameter ratio, D and d are the inner diameter of pipe and the constant-diameter of cone element respectively, C_d is discharge coefficient, ρ is fluid density, A_0 is the opening area of annular channel.

When fluid passes cone element and gradually recovers afterwards, an overall pressure difference δp exists (between pressure taps 1 and 2). δp is caused by the energy loss due to viscous friction and turbulence, i.e. the frictional forces on surfaces mainly due to fluid viscosity and energy dissipation of the turbulence coming from large structures appearing in the shear layers or the recirculating zones [14].

The frictional head loss caused by fluid viscosity occurs along the entire pipeline. It is the energy loss for overcoming the resistive work done by the friction between fluid and pipe wall. The resulted pressure drop depends on pipe diameter, length, roughness, fluid density, specific weight, viscosity, and flow velocity [15].

Fluid flow through valves, orifices, elbows and transitions cause back flow and flow separation which result in the momentum energy loss and the generation or dissipation of turbulent eddies. Especially in DP flow meters where fluid flowing around an obstruction results in the collision between fluid and the obstruction, and even produces severe eddies. For short systems containing many bends, valves, T-junctions and throttling sets, the local head loss is comparable to frictional head loss.

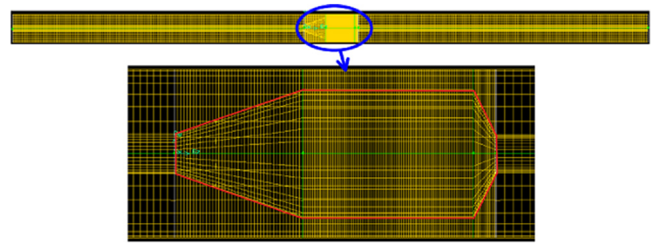

Fig. 2. 3-D mesh of the long-waist cone ($D = 50$ mm, $\beta = 0.65$, $\theta = 40°$ and $\gamma = 60°$).

2.2. Mesh and CFD configuration

The CFD simulations are firstly based on cone elements in Fig. 1 with diameter ratio $\beta = 0.65$, inner diameter of pipe $D = 50$ mm, the front angle of cone element of $\theta = 40°$ and rear angle $\gamma = 60°$, and waist length L varies from 0 to 100 mm, in order to study the flow field within the annular channel of different waist length. The straight pipe upstream and downstream of the cone element is set 10D for flow development as suggested that flow disturbances produced 10D upstream of the cone do not affect the discharge coefficient [11]. A three-dimensional (3-D) mesh is built with GAMBIT as shown in Fig. 2. Subdivide each sharp region into small sections where no sharp edges remained, and then mesh these sub-sections one by one to form matched meshes at the interface between sections. The mesh around the cone element is refined to guarantee precision of the simulation, and the mesh at the center of pipe is also refined to properly match the finer mesh at the top of the cone element. Grid independency has been surveyed by comparing the differential pressures produced with different grid sizes (mesh elements range from 163,000 to 510,000) to make sure the computation converge to a unique solution at an optimum grid size. A medium grid of 325,000 elements is selected to balance computational precision and cost. The commercial CFD software FLUENT is adopted. The range of Reynolds number of simulation is between 14,300 and 74,500. Since the smallest Reynolds number is much higher than the laminar range, the turbulent model must be used in simulations. Several trial simulations with different turbulent models have been done, the results indicate that the k–ε model is more reliable for long and narrow turbulent cases in low Reynolds number range. The RNG k–ε model is currently an appropriate turbulence model in cone flow meter simulations and is thus adopted for this work [16,17]. The pipe wall is processed with standard wall function, and standard pipe roughness height of 2×10^{-5} with roughness constant of 0.5 is selected. Water is chosen as simulation fluid at a temperature of 293 K. Ten different velocity values are tested for the inlet velocity, and the resulting outflow conditions are examined.

2.3. Flow field at the constant-diameter annular channel

A demonstrative 2-D velocity and pressure contours are presented in Fig. 3. The annular channel flattens velocity profile by slowing the incoming flow at the center while increasing it near the wall resulting in a uniform velocity profile [18]. Judging from the pressure contours in Fig. 3(a), the high pressure is better tapped a little further upstream from the cone head in avoidance of possible turbulence and secondary flow (backflow) generated by cone head. Flow separation and reattachment at the entry edge of the constant-diameter annular channel is caused by the incoming fluid separating from cone element when it rushes into the annular channel, and re-attaches to cone surface. It is known as the marginally separated flow regime occurs in a short contraction channel. For a longer traveling path in the annular channel, the flow tends to reattach to cone element in the form of a turbulent boundary layer,