Flow Measurement and Instrumentation 57 (2017) 20–27

Contents lists available at ScienceDirect

Flow Measurement and Instrumentation

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

Algorithm with improved accuracy for real-time measurement of flow rate in open channel systems

Cornelius E. Agu[a,*], Åsmund Hjulstad[b], Geir Elseth[b], Bernt Lie[a]

[a] Faculty of Technology, University College of Southeast Norway, 3918 Porsgrunn, Norway
[b] Statoil ASA, Postboks 8500, 4035 Stavanger, Norway

ARTICLE INFO

Keywords:
Circular channel
Saint Venant equation
Non-Newtonian fluid
Venturi channel

ABSTRACT

Most methods for flow rate measurement in open channels usually have low accuracy over a range of flow rates due to varying fluid properties, flow conditions and channel length. This paper suggests an algorithm to improve on the accuracy of flow rates computed based on hydraulic structure and slope-hydraulic radius methods. A model for determining flow rates in accelerating flows is also developed. In the proposed algorithm, the parameter used for adapting the flow rate *models* is obtained by comparing the measured fluid depth with the depth simulated based on the one-dimensional Saint Venant equations. The results show that an improvement from ± 2.3% to ± 0.8% accuracy in the flow rate measurement using the Venturi flume method could be achieved. In unsteady state flow in a straight-run channel, the results based on flow simulation also show possibility of achieving accurate computation over a wide range of flow rates.

1. Introduction

Fluid flow in an open channel has many industrial applications. It is applied in transportation of slurries, water supply for irrigation, and river flow control [1]. In these fields, accurate flow measurements are important for proper flow distribution and control for safe operations. In open channels, the flow rate is usually difficult to measure directly. Most methods employed are based on computation of flow rate from measurements of other variables that can be measured directly. Such variables include channel width, channel depth, channel slope and channel velocity. Among other methods, the timed gravimetric, the area-velocity, the slope-hydraulic radius and the hydraulic structure methods are used for flow rate measurements in open channels [2].

The timed gravimetric method is limited to flow rates less than100 litres/min and is not suitable for continuous flow. The area-velocity method requires measurement of average velocity of the flow over a known cross-section. The area-velocity method uses pressure transducer and Doppler ultrasonic sensor for depth and velocity measurements, respectively. These instruments are sensitive to flow disturbances, thus resulting in error ± 10% in the measurement [3]. In the slope-hydraulic radius method, a flow resistance model such as the Manning formula is utilized. The method is applied in uniform flows, and is best suited for sizing open channels due to its simplicity. For control purpose, the slope-hydraulic radius method is not suitable due to its wide measurement error in the range of 25–30%. The measurement error is due to uncertainty in determining the correct frictional parameter, such as Manning's roughness coefficient that characterises the flow. Another common method is the use of hydraulic structures such as weirs and flumes. Both structures introduce a restriction in the flow direction, which leads to changes in the approach velocity and in the liquid depth in the channel. The measurement of flow rate with a flume or weir is based on the unique depth-flow rate relationship established in the flow by the structure. Although flumes and weirs show high accuracy (2–6%) under laboratory observations, the field accuracy still lies within ± 10% [4]. This is due to uncertainties in measurement of the level, and due to difficulties in obtaining the correct discharge coefficient for correction of losses in the theoretical depth-flow rate relationship.

This paper focuses on the use of hydraulic structures and slope-hydraulic radius measurement techniques, where the liquid depth is the only physical measurement required to compute the flow rate in a given channel geometry. These techniques are easier to manipulate in designing a software for flow control in open channels. Normally, the hydraulic structures (flumes or weirs) are installed in applications where the flow upstream is subcritical (that is, the flow condition where the flow velocity is less than the gravity wave celerity). When the velocity is greater than the wave speed (celerity), the flow condition is supercritical flow. At the transition between subcritical and supercritical conditions, the flow is critical, that is, the flow velocity and the wave speed are the same. In general, flumes are designed depending on

* Corresponding author.
E-mail addresses: cornelius.e.agu@usn.no (C.E. Agu), ahju@statoil.com (Å. Hjulstad), geiel@statoil.com (G. Elseth), bernt.lie@usn.no (B. Lie).

http://dx.doi.org/10.1016/j.flowmeasinst.2017.08.008
Received 18 April 2017; Received in revised form 1 August 2017; Accepted 9 August 2017
Available online 12 August 2017

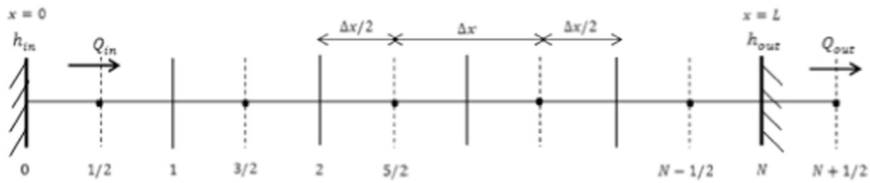

Fig. 1. Computation nodes for liquid depth and flow rate along the channel length.

whether the approaching fluid flow is subcritical or supercritical. Wilson [5] described the design of straight-run channels for measurement of flow rates in supercritical condition. Similar to subcritical Venturi flumes, Kilpatrick et al. [6] and Smith et al. [7] gave clear discussions on development of supercritical flumes. The problems with the use of supercritical Venturi flume are the difficulty to obtain critical flow conditions for all flow rates, and the possibility of deposition of fluid debris or suspensions. These challenges limit the measurement range as well as the hydraulic control of the flume. The slope-hydraulic radius method can be applied in supercritical flow conditions, but this will require in addition to uniform flow model, a model for accelerating flows, since the flow may not have reached a uniform flow before discharging the channel.

There are several studies and model reviews on flow rate measurement in open channels [8–10]. The possibility of estimating drilling mud flow rate for kick/loss detection using a Venturi channel flow rate model is discussed in Berg et al. [11], where it is shown that the required tuning parameter for the model depends on the fluid properties due to non-Newtonian behaviour of the fluid. In this paper, an algorithm is presented for computing flow rate in open channels with improved accuracy. The developed method could be suitable for software implementation in open channels in both subcritical and supercritical upstream flow conditions. The desired improved accuracy is obtained by continuous calibration of the model applied in each of the slope-hydraulic radius and hydraulic structure techniques. In order to achieve this, the flow through the channel is simulated using the estimated flow rate, and the simulated fluid depth is compared with the measured fluid depth. The difference in the simulated and measured depths is used to continuously adjust a tuning parameter in the flow rate model until the difference between the simulated and the measured depths is within a tolerance level.

The success of this algorithm depends on a suitable 1-D unsteady state model that can be applied to simulate the flow in an open channel. The Saint Venant equations have been long established as a good 1-D model that predicts the flow behaviour in an open channel. The accuracy and speed of execution of these hyperbolic partial differential equations depend on the numerical scheme employed. A number of numerical algorithms for solving the Saint Venant equations have been developed [12–14]. The simplified numerical scheme described in Agu et al. [15] for solving the nonlinear equations, is applied in this paper.

In the following sections, the governing equations are presented, and the iterative algorithm for computation of the flow rate using both the hydraulic structure and the slope-hydraulic radius methods, are described. Simulation results based on the algorithms are presented, and their accuracy and speed of execution are discussed. Finally, some conclusions are drawn.

2. Governing equation

The unsteady state flow of fluid in an open channel of any kind of cross section can be described by the one-dimensional Saint Venant equations [16]

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0 \tag{1}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial\left(\frac{\beta Q^2}{A}\right)}{\partial x} = gA(\sin\theta - S_f) - gA\cos\theta\frac{\partial h}{\partial x}. \tag{2}$$

Here, Q is the volumetric flow rate, and A and h are the flow cross sectional area and free surface liquid depth, respectively. θ is the channel angle of inclination, and g is the acceleration due to gravity. β is the momentum correction coefficient with a value between 1.03 and 1.07. x is the position along the channel axis and t is the time. For a Newtonian fluid, the frictional slope S_f is given by Manning's equation

$$S_f = (|V|n_M)^2 R_h^{-4/3}, \tag{3}$$

where n_M is the Manning's roughness coefficient, $V = Q/A$ is the average flow velocity and $R_h = \frac{A}{P_w}$ is the hydraulic radius, where P_w is the wetted perimeter at the flow cross section. In non-Newtonian fluid flows, the internal frictional shearing stresses dominate. Based on the velocity profile for a power law fluid rheology [17], S_f is obtained as given in Eq. (4). For yield-pseudo-plastic fluid rheology, S_f is given by Eq. (5) according to Jin and Fread [18].

$$S_f = \frac{K}{4\rho g R_h}\left(\frac{|V|}{h}\frac{1+2n}{n}\right)^n, \tag{4}$$

$$S_f = \frac{\tau_y}{\rho g R_h}\left[1+\left(\frac{(\epsilon+1)(\epsilon+2)|V|}{(0.74+0.656\epsilon)\left(\frac{\tau_y}{K}\right)^{\epsilon} R_h}\right)^{\frac{1}{\epsilon+0.15}}\right]. \tag{5}$$

Here, ρ, τ_y, K and n (or $\epsilon \equiv \frac{1}{n}$) are fluid properties denoting the density, yield shear stress, flow consistency coefficient and fluid behaviour index, respectively.

The numerical solution of Eqs. (1) and (2) can be obtained as in Agu et al. [15], with notation for the spatial discretization as given in Fig. 1, where the computation nodes for the liquid depth are at the cell centres ($i = 1,2,3, ..,N$) and those for the flow rate are at the cell faces ($i = \frac{3}{2}, \frac{5}{2}, \frac{7}{2}, ..., N+\frac{1}{2}$) based on a staggered grid arrangement. Eqs. (6) and (7) describe the discretized forms of Eqs. (1) and (2).

$$\frac{dA_i}{dt} = -\frac{Q_{i+1/2} - Q_{i-1/2}}{\Delta x}, \tag{6}$$

$$\frac{dQ_{i+1/2}}{dt} = -\beta\frac{(QV)_{i+1} - (QV)_i}{\Delta x} - g\overline{A}_{i+1/2}\cos\theta\,\frac{h_{i+1} - h_i}{\Delta x} + g\overline{A}_{i+1/2}(\sin\theta - S_{fi+1/2}). \tag{7}$$

Applying the first order upwind scheme,

$$(QV)_i = \frac{Q_{i+1/2} + Q_{i-1/2}}{2}V_{i-1/2}$$

$$V_{i-1/2} = \frac{Q_{i-1/2}}{\overline{A}_{i-1/2}},$$

where $\overline{A}_{i+1/2}$ is the average cross sectional area for each cell face, and is calculated based on the average cell centre liquid depth, $\frac{h_{i+1}+h_i}{2}$

2.1. Boundary conditions and inputs

At the upstream boundary, the values of $h(t, x = 0)$ and $Q(t, x = 0)$ are designated as input corresponding to h_0 and $Q_{1/2}$, respectively, as shown in Fig. 1. The downstream boundaries are $h(t, x = L)$ and $Q(t, x = L)$ corresponding to h_N and $Q_{N+1/2}$, respectively. Normally,