Modeling of Conical Central Baffle Flumes Using CFD

Ankur Kapoor, Aniruddha D. Ghare, and Sujith Nair

Abstract A CFD-based simulation study on the Conical Central Baffle Flume (CCBF) is presented in this paper. A CCBF consists of a cone-shaped obstruction positioned vertically at the center of an open channel. Computational Fluid Dynamics (CFD) is a tool to mathematically model a physical phenomenon, numerically analyze and solve the system involving fluid flows. Flow-3D® was used to perform CFD-based three-dimensional numerical simulations of Conical Central Baffle Flumes in rectangular channels. Flow-3D® is a suite of CFD software that provides accurate simulation results because to its simple and effective mesh creation technique and accuracy in capturing free surfaces. A CFD model is developed which replicated the system over which experiments were conducted in laboratory. By comparing the centerline water surface profile of the simulation and experiment flows for the identical discharge and flow parameters, the CFD model developed in Flow-3D® is validated. The comparison shows that the simulation profile, to a great extent, resembles the experimental profile. The simulation flow depths are used to predict discharge using the discharge prediction model for CCBFs with a maximum absolute relative error of 6.02% and mean absolute error of 3.30%. Based on the investigation, the use of the CFD-based numerical analysis is recommended in flow measurement studies using the portable central baffle flumes to recognize the flow patterns and fluid behavior under specific flow conditions. CFD is a useful technique for spotting trends or relative differences linked to changes in the installation, even if it cannot replace a laboratory calibration.

Keywords Conical CBF · Flow measurement · Numerical simulations · CFD · Open channels

A. Kapoor (✉)
Department of Civil Engineering, G H Raisoni Institute of Engineering and Technology, Nagpur 440028, India
e-mail: ankurkapoor06@yahoo.com

A. D. Ghare · S. Nair
Department of Civil Engineering, Visvesvaraya National Institute of Technology, Nagpur 440010, India

P. V. Timbadiya et al. (eds.), *Fluid Mechanics and Hydraulics*, Lecture Notes in Civil Engineering 314, https://doi.org/10.1007/978-981-19-9151-6_11

1 Introduction

Measurement of continuous flows in open channels is essential for optimal water use and proper scheduling for agriculture. The increasing stress and scarcity of water demands a device capable of registering discharge at the desired location and time in an open channel. Such a device will also be of great use even for the metering of water and subsequent billing. Many flow measurement devices have been developed. In open channels, continuous flows are often measured using hydraulic devices that provide a certain stage–discharge relationship. Flume is one such type of hydraulic device usually used in open channels. Various critical depth flumes have been developed and implemented for flow measurement in open channel. The critical depth flumes are generally of fixed type meaning, and the flumes offer measurement of flow at the point of its installation in the channel. The flumes are either constructed by channel side convergence while constructing the channel or fabricated using a metal sheet and fixed at the selected location in the channel. Venturi flume, Parshall flume, cutthroat flume, standing wave flume, etc., are the few fixed type flumes being used for flow measurement in open channels. Though the Venturi and cutthroat flumes are easy to install, the Parshall and standing wave flumes pose certain difficulties as the floor of the flume and the channel are not at the same level. The fixed flumes often tend to settle with time, which needs to be regularly checked for the flume's proper functioning. The accumulation of sediments near the upstream or throat of the flume and the growth of vegetation with time in unlined channels also affect the precision of measurement. Another limitation of the traditional fixed flumes is the inability of a flume to be used at multiple locations since they do not have any movable parts. Since the flume's main objective is to reduce the flow area to cause the occurrence of the critical flow section, which helps to create a unique stage–discharge relationship, it would be helpful to have a low-cost temporary construction to confine the flow area wherever and whenever needed. Such a portable device was first proposed by Hager [4]. The device, then referred to as a mobile Venturi flume, consists of a cylinder placed centrally in a horizontal channel. The availability and streamlined design of the cylinder led to its selection as the obstruction. Hager [3] also developed a variant of the mobile Venturi flume specially designed for circular channels for direct flow measurement by mounting a cylinder at the entrance of the channel having diameter less than that of the circular channel. Hager [3, 4] proposed discharge equations based on the critical energy concept and also provided reference charts between the dimensionless terms consisting of discharge and energy head. The equations related upstream head and discharge in terms of the cylinder and channel geometry.

Samani et al. [9] replaced the graphical approach presented by Hager [3] with a computer model for flow measurement using the circular flume. The objective of the computer model was to calibrate the flume of any size without the need for extensive laboratory experiments. Samani and Magallanez [8] extended the computer model studies for use in trapezoidal channels. The computer model to define the flow through the flume followed the conventional energy and Froude number equations. The governing equations were based on the assumptions of a leveled flume and no energy loss between the upstream and critical section. The study also considered the occurrence of critical flow at the smallest cross-section between the pipe and the channel, which was assumed to be at the center of the flume, and therefore, the critical cross-sectional area was calculated accordingly. Ghare et al. [2] experimentally calibrated the cylindrical flume for use in trapezoidal channels having side slopes ranging from 0.50 H:1 V to 2 H:1 V. The experimental calibration of a measuring device is always based on a limited set of geometric and flow conditions. It is difficult to consider all the possible variations occurring on the field during practical application of the flume, in the physical experiments. The cost of establishing an experimental setup is another factor that often limits the possible change. Excessive manufacturing cost and the challenge of obtaining laboratory data result in the use of CFD to recognize the flow patterns and fluid behavior under specific flow conditions.

Recent advances in hydrodynamics have resulted in the application of numerical analysis based on CFD in flow measurement studies. The use of CFD-based numerical simulation in mobile flumes was investigated by Li et al. [7]. The flow pattern of three cylindrical flumes designed for usage in U-shaped channels was investigated numerically using Flow-3D® software [1]. By comparing the centerline water surface profile of the simulation and physical experiment, the created CFD model was proven to be accurate. One such portable flow measuring device proposed by Kapoor et al. [6] has been investigated using CFD-based numerical model studies in the present study.

2 Conical Central Baffle Flume (CCBF)

The device proposed by Kapoor et al. [6] consists of a cone-shaped baffle (obstruction) placed at the center of a rectangular channel, referred as the Conical Central Baffle Flume (Fig. 1).

An analytical discharge prediction model (Eq. 1) for the conical flume was developed by Kapoor et al. [6], which was based on energy concept and calibrated using

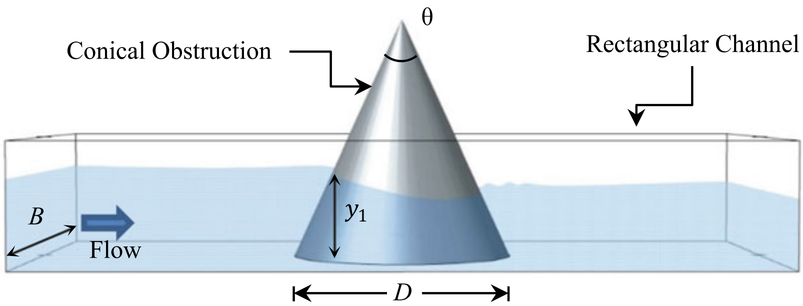

Fig. 1 Conical Central Baffle Flume [6]

laboratory experiments.

$$Q = C_d \sqrt{\frac{\left(B_c y_c + m y_c{}^2\right)^3 g}{B_c + 2 m y_c}} \tag{1}$$

in which $B_c = B - D$ (contracted width) and $m = \tan\frac{\theta}{2}$

The discharge correction factor (C_d) was expressed (Eq. 2) in terms of upstream head (y_1) and contracted channel width (B_c) to correct the unrealistic assumptions of uniform velocity distribution and streamline curvature.

$$C_d = 1.1\left(\frac{y_1}{B_c}\right)^{0.135} \tag{2}$$

The critical flow depth, assumed at the central section of the flume, was expressed as the function of upstream flow depth (y_1) as

$$y_c = \frac{\left(b^2 + 40 m B_c y_1\right)^{1/2} - b}{10m} \tag{3}$$

in which $b = 3B_c - 4my_1$

The proposed discharge prediction model by Kapoor et al. [6] was calibrated using experimental data collected on flumes (Table 1) having different bed widths (B) and cone diameters (D) such that the contraction ratio (D/B) varies from 0.60 to 0.92. The experiments were performed for different discharges such that the dimensionless discharge ($Q_m/B^{\frac{5}{2}} g^{\frac{1}{2}}$) varies between 0.037 and 0.232. The sole physical measurement performed at the cone face after positioning it in the center of the channel is the upstream flow depth (y_1). The proposed model (Eq. 1) predicted discharge with a maximum absolute error of 8.19% and an average absolute error of 4.24%.

In the present study, the CFD-based numerical simulation studies have been conducted on the CCBF proposed by Kapoor et al. [6]. The primary objective of this study is to develop a CFD model for the CCBF and validate it by comparing the simulation results with the experimental data [6]. The validation would ascertain the use of CFD-based numerical analysis in flow measurement studies using the portable central baffle flumes.

Table 1 Dimensions of the tested CCBFs [6]

CCBF	B (m)	D (m)	θ
CCBF-1	0.60	0.360	23.00°
CCBF-2	0.30	0.275	07.63°
CCBF-3	0.30	0.200	30.00°
CCBF-4	0.10	0.080	15.84°
CCBF-5	0.10	0.075	14.77°

3 CFD Simulations

CFD simulations require numerical solutions to the equations controlling fluid flow. Since, water which is a Newtonian fluid is flowing through the flume, the fluid flow could be described by the continuity and momentum conservation equation (Navier–Stokes equations).

For incompressible flows, the mass continuity of the fluid motion is represented in the Cartesian coordinate system as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{4}$$

where u, v, and w are the fluid velocity components in the x, y, and z directions.

The Navier–Stokes equations or momentum conservation equations for three-dimensional flows in the Cartesian coordinate system are represented as

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \tag{5}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{6}$$

$$\begin{aligned}\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)&=-\frac{\partial p}{\partial z}+\rho g_z\\&\quad+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)\end{aligned} \tag{7}$$

where ρ is the density of the fluid, ρ is the pressure, μ is the fluid viscosity, and g_x, g_y, and g_z are the components of body acceleration, whereas u, v, and w are the fluid velocities in the x, y, and z directions, respectively.

The numerical simulations were performed using FLOW-3D® [1] with a one-phase fluid model, free surface interface, and incompressible flow conditions. For free surface modeling, Flow-3D® relies on the Solution Algorithm (SOLA) developed by Hirt and Nichols [5] and the extremely accurate True Volume of Fluid (TruVOF) approach. This method tracks the interface between liquid and gas within the mesh using fluid fraction, ranging from 0 to 1. The novel Fractional Area Volume Obstacle Representation (FAVOR™) meshing technique is used by Flow-3D®. The method allows the construction of simple structured rectangular grids, which makes the mesh easy to generate without affecting the numerical accuracy of the simulation.

In Flow-3D®, a basic simulation model of the CCBF was created that reproduced the experimental flume. The dimensions of the Conical Central Baffle Flumes in rectangular channel are mentioned in Table 1. The CCBF geometry (Fig. 2) was prepared using predefined geometries in Flow-3D®.

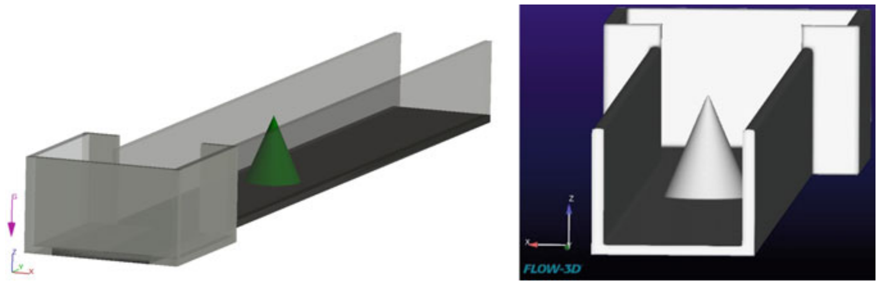

Fig. 2 CCBF geometry prepared in Flow-3D®

The flume geometry was defined using simple structured rectangular mesh which covered the computational domain. Mesh block-1 having 30 mm cell size was generated for the u/s reservoir and the channel. To more precisely capture the flow properties in the area of the obstruction, a finer mesh block-2 with 10 mm resolution was layered within mesh block-1. The boundary conditions specified to the mesh blocks are shown in Fig. 3 in which 'S' denotes *Symmetry*, 'W' denotes *Wall* and 'O' *denotes* Output boundary condition. The RNG k-turbulence model, which is frequently used for external flows with complex geometries in CFD applications, was enabled to simulate the turbulence in the flow. The simulations were conducted until steady-state conditions were reached.

The validation using CFD simulations is performed in two steps. Comparing the water surface profiles of the simulation and experiment flows for the identical discharge and flow parameters is the first step in validating the simulation model setup. After validating the simulated model, the proposed discharge prediction model (Eq. 1) is verified. Similar discharge values to those used in the experiment trials were used in the simulations. The dimensionless discharge $\left(Q^* = \frac{Q}{B^{5/2}g^{1/2}}\right)$ of the experimental runs was found to vary between 0.037 and 0.232.

4 Validation of CFD Model Setup

It is important to first validate the model setup created in the Flow-3D® so that the same can be used for further validation of the discharge prediction model (Eq. 1) and also to perform further simulations for an extended range of flow parameters in the future studies. In the present study, the simulation problem setup is validated by comparing the simulation flow profile with the experimental one for the same discharge. Since the experimental flow depths were recorded along the centerline of the channel therefore, the comparison at various locations was also made along the centerline of the channel by plotting both the water surface profiles for the same discharge values. The comparison of the water surface profiles for CCBF-1 with discharge 42 L per second (lps) is shown in Fig. 4. Similar graphs were plotted

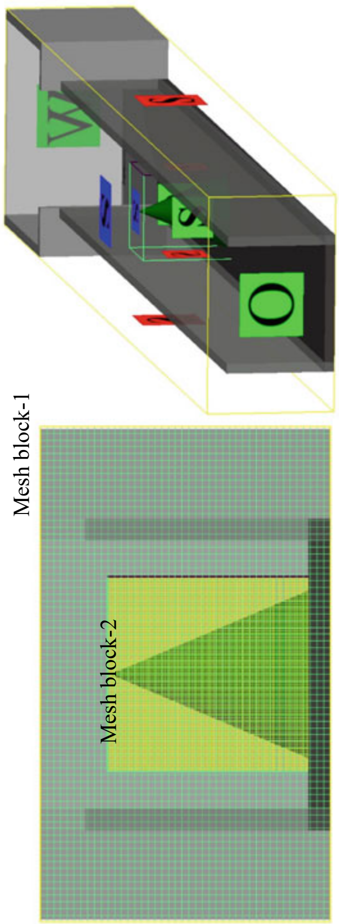

Fig. 3 Mesh blocks and mesh boundary conditions

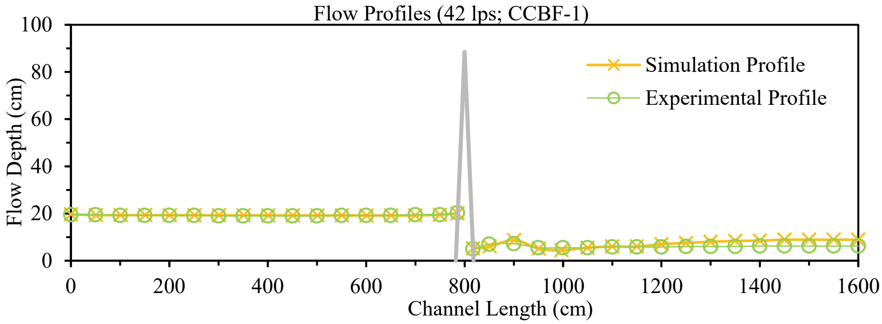

Fig. 4 Flow profile comparison

for all CCBFs and discharges. The comparison reveals that the simulation profile, to a great extent, resembles the experimental profile with an average difference of 2.5% between the two flow profiles. Additionally, comparisons are made between the measured experimental flow depths at the cone face (y_1) for a discharge Q for all the Conical Central Baffle Flumes and the simulation results. The maximum difference between the experimental and simulation flow depths at the cone face (y_1) is found to be 5.81% with a mean value of 2.71%.

Additionally, to comparing the upstream flow depths, the ratio of the simulated flow depth downstream and upstream of the obstacle has also been determined for all of the simulations. This ratio is consistent throughout all simulations and is within the experimentally measured submergence limit range of 0.44 to 0.73 [6]. The comparison of the flow profiles and the simulation submergence ratios certifies the use of created simulation model setup.

5 Validation of Discharge Model Using CFD

Since the simulation and experimental flow profiles closely resembles each other, the simulated upstream depth (y_1) can be used to validate the discharge prediction model developed by Kapoor et al. [6]. The first step toward the model validation is to calculate the critical depth of flow (y_c) using Eq. (3). The equation for the critical depth has been developed assuming that the critical conditions prevail at the central section of the obstruction, observed in the flow direction. Therefore, the calculated critical flow depths (y_c) are compared with the simulation flow depths at the center of the conical obstruction (y_{center}) for all the simulations. It is found that the maximum difference between y_c calculated using Eq. (3) and y_{center} is 4.82% with an absolute mean value of 2.97%.

Thereafter, the calculated y_c has been used to predict discharge using the discharge Eq. (1) for all the simulations and compared with the true discharge values. More than 90% accuracy is obtained in the discharge estimated using discharge Eq. (1) for simulations on the CCBF placed in a rectangular channel (Table 2). The maximum absolute error was found to be 6.02% with an absolute mean value of 3.30%. This shows that the experimentally calibrated discharge Eq. (1) developed by Kapoor et al. [6] works well even on the simulation data and that the CFD-based numerical simulations can be used to ascertain the applicability of the model for more such combinations of flow and geometry parameters.

It is worth mentioning at this stage that the discharge Eq. (1) when applied on the same data based on which it was calibrated gave a maximum error of 8.19% with a mean value of 4.24%; however, the error values were significantly reduced when applied on the simulation data. In addition to this, the percentage observations giving less than 5% error in discharge prediction were 53% in case of experimental

Table 2 Discharge calculations for simulation data

CCBF#	y_1 (cm) [Simulation]	y_c (cm) (Eq. 3)	C_d (Eq. 2)	Q (lps) (Eq. 1)	Q_m (lps)	Error (%) $\left\lvert \frac{[6]-[5]}{[6]} \right\rvert$	Dimensionless discharge (Q^*)
[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]
CCBF-1	16.97	11.64	1.050	32.95	32.00	2.96	0.037
	19.99	14.00	1.073	43.75	42.00	4.17	0.048
	22.42	16.03	1.090	53.45	52.00	2.79	0.059
CCBF-2	10.37	07.50	1.105	07.49	07.20	4.01	0.047
	11.55	08.50	1.122	09.10	08.80	3.37	0.057
	12.49	09.20	1.134	10.49	10.00	4.89	0.065
	13.00	09.40	1.140	11.29	10.80	4.49	0.070
	14.00	10.30	1.151	12.93	13.00	0.52	0.084
CCBF-3	18.10	13.20	1.437	06.29	06.20	1.48	0.040
	18.99	13.90	1.446	06.89	07.20	4.28	0.046
	23.50	17.50	1.489	10.38	10.00	3.84	0.065
	25.00	18.70	1.501	11.72	11.80	0.69	0.076
	26.50	19.50	1.513	13.14	13.00	1.10	0.084
CCBF-4	07.08	05.20	1.305	01.12	01.06	6.02	0.107
	08.01	06.00	1.327	01.42	01.36	4.58	0.137
	08.61	06.50	1.340	01.63	01.56	4.80	0.157
	09.38	07.00	1.355	01.93	01.86	3.85	0.188
	10.27	07.70	1.372	02.31	02.30	0.37	0.232
CCBF-5	07.25	05.10	1.270	01.33	01.28	3.90	0.129
	07.83	05.82	1.283	01.53	01.48	3.62	0.149
	08.03	05.95	1.288	01.61	01.57	2.37	0.158
	08.47	06.20	1.297	01.78	01.72	3.24	0.174
	08.92	06.60	1.306	01.96	01.87	4.66	0.189

Fig. 5 Frequency distribution of errors associated with experimental and simulation data

data, whereas the same were found to be nearly 96% in case of simulation data. The frequency distribution of relative absolute errors in discharge prediction associated with both experimental and simulation data is shown in Fig. 5.

6 Conclusions

The Conical Central Baffle Flume has been investigated using CFD. Flow-3D® has been used to perform the numerical simulations. The simulations were performed for those CCBFs and discharges over which the actual experiments were conducted. Therefore, the CFD model was validated using the experimental data by comparing the simulation and experimental flow profiles. The simulation flow profiles resemble the experimental profiles validating the CFD model. The average difference between the experimental and simulation flow depths at the cone face (y_1) is found to be 2.71%. The upstream flow depths (y_1) were used to calculate the critical depths (y_c) and compared with the simulation flow depths at the center (y_{center}). The calculated y_c was then used to predict discharge using the discharge Eq. (1) for all the simulations and compared with the actual discharge values. The average absolute error in discharge prediction was found to be 3.30%. Additionally, the ratio of the downstream and upstream simulation flow depths was found to lie between 0.44 and 0.73, which corresponds to the submergence limit recorded during experiments. Based on the comparisons between y_c and y_{center}, , it can be concluded that as long as the submergence is less than 73%, the proposed equation for critical depth (Eq. 3) may estimate y_c at the center with an accuracy of over 95%. Moreover, it has been discovered that when the submergence ratio is less than 0.73, the discharge estimated using discharge Eq. (1) for simulations on the CCBF placed in a rectangular channel is more than 90% accurate. The results of the present study define the accuracy and capability of CFD to capture actual flow scenarios. The CFD model can be used to simulate flow through CCBFs for an extended range of flow parameters in the future studies.

References

1. FLOW-3D® Version 12.0 [Computer software] (2019) Santa Fe, NM: Flow Science, Inc. https://www.flow3d.com
2. Ghare AD, Kapoor A, Badar AM (2020) Cylindrical central baffle flume for flow measurements in open channels. J Irrig Drain, ASCE 146(9):06020007. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001499
3. Hager WH (1988) Mobile flume for circular channel. J Irrig Drain, ASCE 114(3):520–534. https://doi.org/10.1061/(ASCE)0733-9437(1988)114:3(520)
4. Hager WH (1985) Modified venturi channel. J Irrig Drain, ASCE 111(1):19–35. https://doi.org/10.1061/(ASCE)0733-9437(1985)111:1(19)
5. Hirth C, Nichols B (1981) Volume of fluid (VOF) method for the dynamics of free boundaries. J Comput Phys 39(1):201–225
6. Kapoor A, Ghare AD, Vasudeo AD, Badar AM (2019) Channel flow measurement using portable conical central baffle. J Irrig Drain, ASCE 145(11):06019010. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001427
7. Li X, Jin L, Bernie AE, Yang Z, Wang W, He W, Wang Y (2020) Influence of the structure of cylindrical mobile flumes on hydraulic performance characteristics in U-shaped channels. Flow Meas Instrum 72. https://doi.org/10.1016/j.flowmeasinst.2020.101708
8. Samani Z, Magallanez H (1993) Measuring water in trapezoidal canals. J Irrig Drain, ASCE 119(1):181–186. https://doi.org/10.1061/(ASCE)0733-9437(1993)119:1(181)
9. Samani Z, Jorat S, Yousaf M (1991) Hydraulic characteristics of a circular flume. J Irrig Drain, ASCE 117(4):558–566. https://doi.org/10.1061/(ASCE)0733-9437(1991)117:4(558)