International Journal of Scientific & Engineering Research, Volume 5, Issue 7, July-2014
ISSN 2229-5518

CFD Simulation of Baffle System to Tranquillise the Flow from a Sluice

Geethu G, N. Sajikumar
Government Engineering College Trichur
Thrissur,India
surabhigeethus@gmail.com, saji@gectcr.ac.in

Abstract— **Flow control structures are inevitable structures in a canal/ river system which is used to control the water levels and flow discharge in rivers, open channels and waterways. Sluice gates and Baffle systems fall under such category. Flow behavior through sluice and baffle systems should be analyzed for proper controlling of water flow through the channels. CFD simulation of flow through such structures is very helpful in visualizing the various effects of low. Present study includes the numerical simulation of flow through sluice and baffle systems to study the tranquillising capacity of baffles by using CFD software. The study aims at investigating the effect of sluice gate and vertical baffles. To simulate 3D incompressible viscous two- phase flow, the volume of fluid (VOF) method based on the finite volume method has been utilized. The modelling is done by using pressure based solver. It is found that a baffle of 0.3m size is more effective in tranquillising the flow from the sluice.**

Keywords— CFD; Baffles; Volume of Fluid

I. INTRODUCTION

Flow control structures have been used to control the water levels and discharge in rivers, open channels and waterways. These are inevitable structures in the canal system of an irrigation system. The regulation and distribution of flow through a hydraulic structure on an irrigation system is, to a large extent, controlled by gates of various types. Type of gates that is required for a particular purpose depends on many factors like the dimensions of the gate, the water pressure, the mode of operation and on the availability and cost of local materials [1]. Sluice gates are one of the several such structures. Low efficiency in the water distribution of irrigation networks is generally due to poor performance and leads to very high losses of water in agriculture sector. Since proper use and appropriate operation of these structures has made network management possible, water allocation on time and reduced losses, the identified deficiencies and structural problems in the selection, design, construction, installation and operation of networks will greatly help to prevent water loss and thereby increase the efficiency[1]. Experimental or manual performance evaluation of these systems is tedious and costly. Hence, application of numerical modelling to flow simulation in hydraulic structures is a viable method for the performance evaluation of these structures.

.

Though the steady state of flow condition is customarily used for the design of various flow controlling devices, these structures most often works in the unsteady flow conditions. Simulation of unsteady flow can provide the variation of water surface elevation and discharge in the canal with respect to time. However, such simulations of unsteady systems have limited widespread use, owing the complexity in the analysis. Hydrodynamic models are very much suited to simulate the flow through the network, to evaluate network performance and to monitor hydraulic performance of structures [1].The laboratory experiments on physical models of flow phenomena which have interactions with various types of hydraulic structures may be expensive and time consuming [2]. Limitation in theoretical and experimental investigations has made computational fluid dynamics (CFD) the major means of modelling free-surface motions and exploring free surface physical processes. Simulation of flow through irrigation structures using CFD is one of flow analysis methods which is developed in recent years and is capable of providing precise information regarding the flow. In this study, flow through baffles and sluice gates are simulated using CFD for assessing the ability of the baffle system in a sluice to tranquillize the flow through the system. ANSYS Fluent is used for the CFD modelling of flow through baffle and sluice systems.

II. LITERATURE REVIEW

The discharge through irrigation canals is commonly controlled by means of sluice gates. Many studies were carried out to formulate both free and submerged flow conditions primarily for vertical sluice gates and radial gates. In the first extensive experimental study, a useful diagram for discharge coefficient in free and submerged flow conditions was developed [3]. These results were then reaffirmed in [4]. A study about hydraulic characteristics and discharge control of sluice gates were also done [5]. They investigated various characteristics of a vertical sluice gate in a rectangular flatbed channel. The flow behavior in submerged radial gates was also studied [6]. Four rectangular sluice gates were calibrated for submerged flow conditions. A dimensional analysis was applied to investigate both free- and submerged-flow conditions through a radial gate [7]. Recently, many researchers have carried out different numerical techniques including that of finite volume to solve the Reynolds time-averaged Navier-Stokes equations [8]. A study about the validity of Reynolds Average Navier–Stokes (RANS) simulations for sluice gates in free flow were conducted and the study were focused on the influence of pressure field and mesh size on accuracy of simulation [9], [2]. However, limited number of studies is available which models the baffles system in a channel in general and no such studies in the baffles system in front of sluice in particular. Hence, the current study envisages using CFD for analyzing the flow in a baffle system in front of a sluice to tranquillize the flow.

III. METHODOLOGY

A. Principles of Computaional Fluid Dynamics

In physics, fluid dynamics is a sub part of fluid mechanics that deals with movement of fluid in flow domain. The basic principles of Fluid dynamics lie on the laws of nature, viz., conservation of mass, energy and momentum. Conservation law states that a particular measurable property of an isolated physical system does not change as the system evolves. The conservation of a certain flow quantity means that its total variation inside an arbitrary volume can be expressed as the net effect of the amount of the quantity being transported across the boundary, any internal forces and sources, and external forces acting on the volume [10]. Transport of a conserved quantity can be mathematically expressed by using continuity equation on that quantity. In general, two conservation equations are used which are

- The conservation of mass
- The conservation of momentum

The mass conservation law states that mass cannot be created in a system, nor can be destroyed. This is applied for single phase fluids. The equation for conservation of mass or continuity equation can be written as [11],

— (1)

This is the general form of mass conservation equation and is valid for incompressible and compressible fluids. The source S_m is the mass added to the continuous phase from the dispersed phase.

The equation for momentum conservation can be written as [11],

— (2)

Where p is the static pressure, ρg and F are gravitational and external body force and is the stress tensor.

Before solving the fluid flow problems, CFD model needs the domain in which the fluid flow takes place has to be discretized into smaller control volumes (rather cells in which the aforementioned conservation laws are applied). The software has so many tools for creating a geometric shape, from basic shapes such as a rectangle to more complex shapes. Theses shapes then need to be meshed to form a grid of triangles or quadrilaterals (prism or tetrahedron in case of 3D). The grids can be considered as structured or unstructured. Structured grid is a grid, composed of points in a regular pattern. These are usually created with quadrilaterals or prisms. A grid having irregular pattern of points is called an unstructured grid and is usually created from triangles or tetrahedrons [12].

IV. SIMULATION

The model consists of a rectangular block and two surfaces. The basic geometry is shown schematically in Fig. 1. The geometry is made such that flow from a sluice can be simulated. Here, the tank is partitioned into two parts and the flow from the one part moves to the other part through a sluice. The sluice baffle system is modelled using the porous jump boundary condition. Discrete baffle blocks are provided in the right tank bottom. The flow is modelled over various heights of baffle blocks. 0.3m, 0.5m and 1m are the different heights of baffle blocks. Figure below shows the baffle system of height 0.5m. A continuous baffle of height 0.3m and extend to the full width is provided as another case. For modelling of water flow, the geometry must be meshed and then exported to ANSYS Fluent. Meshed geometry is shown in Fig. 2. Hexahedron meshes are used for making grids.

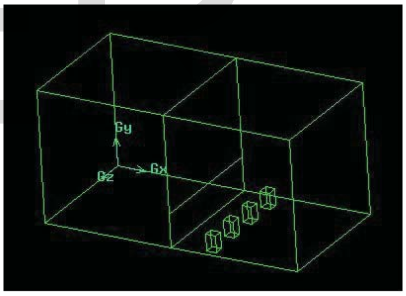

Fig. 1 Problem specification

After exporting to Fluent, grid check of geometry is carried out. The mesh checking option in the software examines various aspects of the mesh, such as the mesh topology, boundaries, counters, and node position with respect to the axis, and provides a mesh check report with details about domain extents, statistics related to cell volume and face area, and information about any problems associated with the mesh.

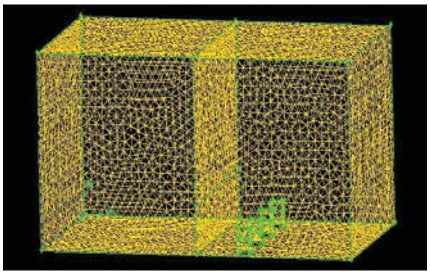

Fig. 2 Meshed geometry with baffles

Since the flow involved in this project is incompressible, unsteady, turbulent and multiphase, a pressure based solver with unsteady flow modeling enabled, k-ε turbulent model and volume of fluid methods are respectively selected.
Sluice gates are modelled as porous jump in this study. The inputs for porous jump are pressure jump coefficient, permeability and thickness. In order to have a fully opened 100% opening for the porous jump was selected. Hence the necessary parameters were adjusted in that manner. The thickness of porous jump for the current study is selected as 0.003m. The parameters required for porous jump are as shown in the Table.1. These parameters have been calculated based on modelling requirements of porous jump boundary condition of the software.

TABLE I. VALUES OF PARAMETERS

Percentage of opening	**Inverse of Permeability m^2**	**Pressure jump coefficient 1/ m**
100	1.0	170.05

V . RESULTS AND DISCUSSION

As indicated earlier, it is necessary to model flow through the baffle system in a sluice using CFD. The flow was simulated by assigning various boundary conditions and flow conditions. Typical flow simulation stages from the phase animation are given in Fig.3. The figure gives the nine stages of the animated phase simulation. It is interesting to note that the CFD simulation can capture the effect of all physical processes- spilling, rolling and other natural phenomena. This figure shows stages of water surface profile when sluice is opened, when the water touches the other wall, when the water rolls back and the like.

In order to assess the tranquillizing effect of baffles, these were provided in front of the sluice. Here remodelling of the geometry had to be carried out to incorporate the baffles in the model. Four types of baffles are considered. In the first type three discrete baffle blocks of dimension 0.3m x 0.3m x 0.3m were used and the flow was simulated. In the second type, a baffle size of 0.3m x 0.5m x 0.3m was used again with three baffles.

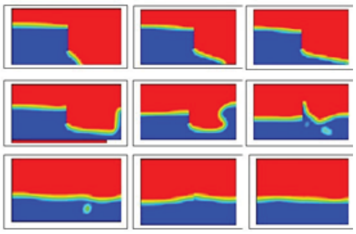

Fig. 3 Stage of the animated phase profiles (9 stages)

A baffle size of 0.3m x 1m x 0.3m and one continuous baffle of dimension 0.3m x 0.3mx 4m were used in third and fourth types respectively. In all the first three types, four number of discrete blocks were used. The flow over these baffles was analyzed using Fluent and the tranquillizing effect of each baffle was analyzed. Various stages of flow in these cases are depicted in Fig.4.to Fig.6. It may be noted that these figures are the screen shots from various stages of animation of the flow. Hence, one can get complete physical feeling how the flow behaves.

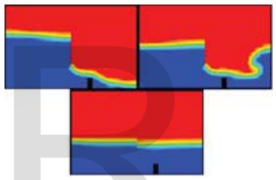

Fig. 4 Flow over baffle of height 0.3m

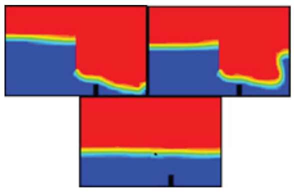

Fig. 5 Flow over baffle of 0.5m height

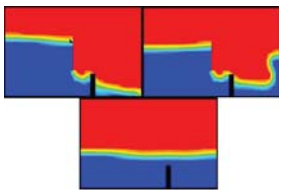

Fig. 6 Flow over baffle of 1m height

Table 2 shows the time taken to reach the water on various levels. Details of flow in the tank without baffle are also included in the table. On comparing the results of additional baffle system with that without such baffles, the time taken to reach the stream of water to various levels are less than that without a baffle. If a baffle is provided on the flow path, it can reduce the sloshing of water and there by obtain a steady state within fewer seconds. If a baffle is not provided in flow path, it takes more time to attain steady level. The more height can increase the time taken to reach steady level. Hence a 0.3X0.3X0.3m is the best arrangement among the tested configurations.

Result shows that, the time taken by the stream of water to reach various levels in the case of continuous baffles is higher than that of the discrete baffles. Because the continuous baffles create more obstruction to flow than discrete baffles. As the time taken to touch the water on right wall increases, the time taken to attain steady level also increases. Hence a continuous baffle is not effective in tranquillising the flow. The times taken to reach the water on different levels are almost same for the discrete baffles. For baffle with more height, the time taken is little more than other discrete baffles. As the height of baffle increases, the obstruction to the flow also increases. Hence a discrete baffle system of 0.3X0.3X0.3 m is ideal one among the all the cases tested.

TABLE II TIME TAKEN BY WATER OVER DIFFERENT BAFFLES

Baffle height (m)	**Water touches on right side wall (s)**	**Water starts to flow backwards (s)**	**Steady level (s)**
0.3	0.77	1.55	7.18
0.5	0.77	1.56	7.18
1.0	0.79	1.58	7.20
Continuous baffle	0.99	1.81	7.56
Without Baffles	0.76	1.78	10.50

VI. CONCLUSION

In this study, the effect of various baffle configurations in a sluice system for tranquillizing the flow is assessed by means of CFD simulation. Based on the study, the following conclusions were made:

1. CFD simulation is an effective method to study the tranquillising effect of different configurations of baffle systems provided in the sluice of channels.
2. A continuous baffle that is provided in front of the sluice is not effective in the tranquillising the flow from the sluice
3. Discrete baffles is capable inducing tranquillising effect in the flow. However, the height has a negative effect over tranquillising though small. Hence 0.3m x 0.3m x 0.3m can be considered as the best option among the four types of baffle tested.

References

[1] Seyedjavad M.S., Mashaal M., Montazar A. (2013) : ' Evaluation of Hydraulic Sensitivity Indicators for Baffle Modules (Case Study: Varamin Irrigation and Drainage Network)', *Journal of hydraulic structures*, pp.33-43.

[2] Akoz M.S., Kirgoz M.S., Oner A.A. (2012): ' Experimental and numerical modelling of a sluice gate flow', *Journal of hydraulic research*,vol.47,pp.167-176.

[3] Henry H.R. (1950): 'Diffusion of submerged jets', ASCE Transactions, Paper No. 2409, Vol.115, pp. 688-693.

[4] Rajaratnam N., and Subramanya K. (1967): "Flow equation for the sluice gate.", *Journal of Irrigation and Drainage Engineering*, Vol. 93,pp. 167–186.

[5] Yen J., Lin C.H., and Tsai T.C. (2001): ' Hydraulic characteristics and discharge control of sluice gates', *Journal of the Chinese Institute of Engineers*, Vol. 24, No. 3, pp. 301-310.

[6] Clemmens A.J., Merkley G.P., Mateos L., Lozano D.(2009): 'Field calibration of submerged sluice gates in irrigation canal', *Journal of irrigation and engineering*, Vol.135, pp.763-772.

[7] Bijankhan M., Ferro V., Kouchakzadeh S. (2013): 'New Stage-Discharge Relationships for Radial Gates', *Journal of Irrigation and Drainage Engineering*, Vol. 139, No. 5.

[8] Cassan L., and Belaud G. (2012) : ' Experimental and Numerical Investigation of Flow under Sluice Gates', *Journal of Hydraulic Engineering*, vol. 138, pp. 367-373.

[9] Kim D.G.(2007): ' Numerical analysis of free flow past a sluice gate', *KSCE Journale of civil engineering*, Vol.11, pp.127-132.

[10] Blazek J.(2001) : 'Computational Fluid dynamics: principles and applications', Elsevier science ltd.

[11] Fluent, 2006, ANSYS FLUENT Theory Guide, ANSYS Inc., Lebanon

[12] Inwood S.(2004) : 'Using Computational Fluid Dynamics to Solve Fluid Flow Problems', Project Report, Laboratory for product and process design, University of Illinois at Chicago.