CFD Simulation of Velocity Distribution in a River with a Bend Cross Section and a Cubic Bed Roughness Shape

Hayder Q. Majeed *, Ali M. Ghazal **

* Lecturer, Ph.D., University of Baghdad/ College of Engineering, Water Resources Dept., Baghdad-Iraq, h.majeed@coeng.uobaghdad.edu.iq.

** Ministry of Water Resources / National Centre for Water Resources Management, alimezaal4@gmail.com

Abstract: The variation of bed roughness along a channel cross section especially if the cross section with bend affects the velocities in that part of the section may be used to change the flow patterns in the channel and as a mean of river training. In this research the Computational Fluid Dynamics (CFD) for modelling the effect of bed roughness on the free surface flows in open channel has been validated for a straight river, the effect of angles of attack downstream the bed roughness has been investigated. This study presents the results and analysis of numerical simulations that were carried out to compute Manning's coefficient of artificial geometric roughness element (cubic shape). Artificial roughness elements were fixed on bed of the flume within the test section in different spacing and angles of attack. A sequence of CFD simulations using the k-ω SST model are compared with a set of laboratory data for the influence of flow at a free surface in open channel with cubic shape bed roughness. The numerical results gaining confidence in a modelling of Volume of Fluid (VOF) technique and allows for modelling of more complex hydraulic structures by including the effect of angles of attack to the simulation.

Keywords: Manning's coefficient, angle of attack, bed roughness, CFD.

1. Introduction

Hydraulic calculations to estimate flow in rivers need to know the roughness characteristics of these rivers. Knowledge of flow resistance which is denoted as Manning's roughness coefficient for different flow conditions aids to construct well water running systems. By way of Computational Fluid Dynamics (CFD) discovers request in an expanding amount of logical and designing functions, the approval of this product is regularly accepted by the operator. In any case, presently that CFD software has the ability to illuminate a tremendous scope of engineering issues, the approval substantial that goes with them can just ever apply to a subset of these applications. The latter are required to accept that the product, whenever utilized accurately, gives results that can be depended on. This absence of approval material is most prominent in application territories that are new to the utilization of CFD. Thus, there
are regularly noteworthy and alarming holes in the approval of CFD. One such application area is free surface simulation in hydraulic structures for example open channel flow with artificial bed roughness. There are a few proofs to legitimize the utilization of CFD in these areas. The motivation of the present CFD modelling relates to study the flow properties for a particular geometry and case of the artificial bed roughness (cubic shape) located in meandering river, since the artificial roughness into a river channel is one of the methods of changing the flow patterns in a river channel.

2. Literature Review

Saeed (1) Studied the hydraulic resistance including Chezy's resistance coefficient, friction factor and Manning's roughness coefficient and velocity distribution over the entire depth of flow for selected several models has been recorded and well fitted by a semi logarithmic equation in artificially roughened steep open channels. He used four models were constructed to simulate channels of 0.1 slopes, in 10 m long and 0.3 m wide laboratory flume. The first model was roughened with continuous prismatic elements of 2cm base width and 1cm height. The second model was roughened with prismatic elements of 2cm base width, 1cm height and 2cm length. The third model had a bed roughened with continuous prismatic element of 3cm base width and 1cm height. While, the fourth model was roughened with prismatic elements of 3cm base width, 1cm height and 3cm length. In each model different intensities roughness elements have been tested. The results of laboratory experiments was the relative velocity \( \frac{v}{v_f} \) decreases to its minimum value at roughness intensity \( \lambda_r = 0.125 \) where \( \lambda_r \) is roughness intensity defined by the ratio of the sum of projected area of roughness elements normal to mean direction of fluid movement to the total floor area (non-dimensional parameter), the longitudinal spacing between roughness elements included in this perimeter. The maximum bed resistance was examined for constant relative depths and found to occur intensity \( \lambda_r = 0.125 \). For all models tested in this study, value of Von Karman coefficient and equivalent sand roughness are obtained. The variation of manning's roughness coefficient with roughness intensity is obtained showing that continuous prismatic elements with 0.125 intensity offer maximum resistance to overtopping flow.

Graf and Blanckaert (2) This paper summarizes some of the main experimental findings. It mainly presents on an experimental study of the flow around a river bend over a mobile bed. Itemized estimations have been made on a fine grid in the external portion of the cross-section at 60º into the curve. Spatial distributions of the mean downstream velocity, the cross-stream motion beside the mean-flow and turbulent kinetic energy are displayed. The downstream velocity increases in outward direction and the core of maximum velocities is in the lower section of the water profile close to the outer bank. As compared with open channel with a straight flow, a considerable reduction of the turbulence activity is observed in the outer-half of the cross section. In the study of flow in river bends, often simplified flow equations have been adopted, which are not justified in the light of the here-reported experimental findings. A more complete system of simplified flow equations has been proposed and solved.

Pradhan, Kumar Khatua (3) considered that the study the distribution of velocity and flow in a river with bends is a significant to be inspected from a practical point of view. When flow reaches a bend, the centrifugal force increasing from the channel curvature due to a horizontal slope in the river level. In their work, the experiential run is made for two different bed roughness sinuosity meandering channel with angle of attack equal to 110º. The target of
the investigation is to discover the impact of curvature and roughness on the flow profile, during the meandering river. It is found that the confrontation of flow, on the flatter bed channel, is larger than that of the channel with higher Manning’s $n$ above a roughness surface.

2. Experimental Studies
The measurements by (4) were examined in a horizontal slope flume of length of 12.5m with cross-section 0.3m wide and 0.45m deep. The cubic artificial roughness element with dimensions of (3x3x3cm) from the physical model was picked in the examination. The roughness element is made of wood and paint with has been making elements roughness of the wood was painted with insulating dyes and so not to change size upon wetting and to protect them from damage by the water Figure (1). The roughness elements were fixed to channel bottom using special adhesive use (Silicone pump). Different configurations of artificial roughness elements were examined to investigate the performance of different types of artificial roughness elements in dissipating of the energy of the flow In order to calculate the Manning’s. Different configurations were achieved by varying the spacing between the elements in each row, the spacing between the rows of the elements, and the location of the rows.

Figure (1): Cubic roughness element
Three configurations were designed as shown by Figure (2). The details of these configurations are shown in Table (1). The spacing between elements in the rows of in configuration number 1, 2 and 3 were 3cm. The spacing between the rows of the elements of configuration 1 is 3.94cm, of configuration 2 is 2cm, of configuration 3 is 4cm.

Figure (2): Types of configurations 1, 2 and 3 of cubic roughness elements

| Configuration | Spacing Between Elements (cm) | Spacing Between Rows of Elements (cm) |
|---------------|-------------------------------|--------------------------------------|
| 1             | 3                             | 3.94                                 |
| 2             | 3                             | 2                                    |
| 3             | 3                             | 4                                    |

Table (1). Details of the configurations of the cubic roughness elements on the test section
| Configuration number | Number of row | Spacing between elements in row, cm | Spacing between rows, cm | 𝑨_𝑑 (areal density) | Manning Coefficient (n) |
|----------------------|---------------|-----------------------------------|--------------------------|---------------------|------------------------|
| 1                    | 15            | 3                                 | 3.93                     | 0.225               | 0.053                  |
| 2                    | 10            | 3                                 | 7.78                     | 0.15                | 0.050                  |
| 3                    | 6             | 3                                 | 16.4                     | 0.09                | 0.047                  |
| Without roughness element | -   | -                                 | -                        | -                   | 0.017                  |

3. CFD Model—Theory
The solution of the Navier-Stokes equations are considered for the numerical modelling in this study, which are relay on the conservation of momentum and mass assumptions inside a fluid. The mass conservation is defined by the differential equation:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \]

where \( \mathbf{v} \) is the velocity of the fluid and \( \rho \) is the density. The conservation of momentum is similarly presented below:

\[ \frac{\partial}{\partial t} (\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \tau \]

where \( \tau \) is the stress tensor and \( p \) is the pressure there is an additional transport equation is used to describe the properties of flow turbulent on the flow. These extra transport equations are resolved for several quantities of turbulence. The Volume of Fluid (VOF) procedure by Hirt and Nichols (5) is considered. In this method, the coupling is presented by defining the volume fraction, \( \alpha_i \), where \( i \) denotes to the phase. The volume fraction for the \( i \)th phase is the volume fraction of a cell inside that phase. Once modelling the free surface between air and water, the equation of transport is used for the water phase,

\[ \frac{\partial \alpha_w}{\partial t} + \nabla \cdot (\mathbf{v} \alpha_w) = 0 \]

where \( \alpha_w \) is the volume fraction of water. The volume fraction of the other phase can be inferred from the constraint,

\[ \alpha_a = 1 - \alpha_w \]

where \( \alpha_a \) is the air fraction of the volume. The Finite Volume method is considered to resolve the equations 1 to 4 and depends on how can be divided the flow into a grid contain of many cells. If the cell consists of just water, then \( \alpha_w = 1 \); if not, then \( \alpha_w = 0 \). For cells which contain both air and water, \( 0 < \alpha_w < 1 \). In each cell the fluid characteristics are modified with respect to the specified volume fraction, i.e., the cell density is,

\[ \rho = \alpha_w \rho_w + (1 - \alpha_w) \rho_a \]

The typical conditions for the boundary accessible in CFD programming have their inspiration in aeronautic and mechanical design however can be utilized astutely to demonstrate upstream and downstream limits found in open channels. This can include unphysical alterations to the area: for instance, the presentation of an even mass flow inlet in front of the impediment, as utilized by Sarker and Rhodes (6). In any case, CFD programming presently incorporates boundary conditions that are explicit to the open channel.
circumstance, at which the downstream and upstream water elevation can be determined. At a vertical upstream boundary, a pressure inlet is utilized at which the entire pressure, $p_0$, is given by:

$$p_0 = \frac{1}{2} (\rho - \rho_a) V^2 - (\rho - \rho_a) g (y - h_0) \quad \text{-------------------------------------} (6)$$

At height, $y$, $\rho$ is the density of the phase, where $V$ is the velocity and $h_0$ is the measured height, and. The hydrostatic pressure at each point on the inlet, is determined dependent on the relative elevations of the free surface and the point and on the fluid density neighbouring for that point. The density beneath the constant free surface is thought to be that of water, and overhead it considered the air density. A comparative methodology is utilized at the boundary of the downstream flow.

4. CFD Simulation

4.1 Solver Setup

The models presented here are done using FLUENT (V17.0). For the validation study, open channel flume 3D simulation was applied, with a width of 0.3 m. With a 3D model it is conceivable to find a mesh that resolves the vertical, streamwise and transverse directions with adequate precision. The standard wall functions k-\(\omega\) SST turbulence model of Menter (7) was utilized. This procedure of turbulent models categorised as Reynolds-Averaged Navier-Stokes (RANS) models. These simulations, with time-averaged approximations, are broadly utilized in modern applications. The k-\(\omega\) SST has recognised favourable circumstances in case of a strong bend in the streamlines, for instance, flow in the meandering rivers. To finish depiction of the CFD demonstrating, due to gravity, the body force-weighted pressure discretization scheme was considered; discretization with second-order schemes were applied for turbulence kinetic energy and the momentum, and dissipation equations; and for pressure velocity coupling algorithm the Pressure Implicit with Splitting of Operator (PISO) was proposed, merely because it is considered specifically for transient simulations (8). A time step of 0.01s was utilized all through to keep the simulation stable due to the requests of the VOF model. the flow fields being modelled for 65s to establish, it tends to be acknowledged what number of time-steps are needed to arrive at consistent state conditions.

4.2 Domain Geometry and Boundary Conditions

CFD modelling of the artificial bed roughness proposed by Ghazal (9) is shown in Figure (1). This artificial cubic bed roughness has been simulated in CFD model and as shown in Figure (3). The boundary conditions of numerical simulation are set-up to matches with the experiment conditions has been indicated in Figure (4), the water flow depth is also shown in the figure. At the inlet of the domain a pressure velocity was considered. When displaying a free surface in FLUENT, it should determine the free surface height, comparative with the datum. Internally calculation found the static pressure and the volume fraction at the inlet dependent on the situation of the face, comparative with the free surface situation as appeared in Equation (6). The energy head likewise required to consider the flow dynamic pressure. At the downstream pressure outlet, just the bottom height level was required. The boundary overhead the air phase was considered as a symmetry condition, which results as a zero-shear stress and zero velocity. Utilization of a symmetry boundary condition along these areas is a typical practice for such case, i.e., open boundaries. The rest of boundaries are considered as walls, which is no-slip condition was applied.
4.3 Mesh Generations

Hexahedral mesh is considered in this study as shown in Figure (5). This is since the efficient solution of the prism layer in hybrid mesh. The viability of prism layer gets a more precise outcome also showed in past studies (8). The entire number of elements after the refinement is about 650000 with maximum skewness is approximately 0.85, which is less than 1.0. Additional improvement is considered close to the wall domain. The grid cell size close to the walls (value of $y^+$) is significant and the $y^+$ value relay on the modelling method applied. k-ω SST model is considered in this investigation, to simulate the turbulence condition. In this study, the maximum $y^+$ value of 16.4 are generated close to wall grid
resolution and as shown in Figure (6) and (10). The maximum grid generation with near wall meshes has a big effect on model efficiency.

![Hexahedral mesh generation](image)

**Figure (5):** Hexahedral mesh generation.

**Figure (6):** Maximum y⁺ values close to the bed.

### 4.3.1 Mesh Independence Study

The mesh utilized in CFD can dramatically affect the precision of the solution. In this study there were three grid of mesh sizes (325,000; 650,000 and 1,300,000 mesh) and as shown in Figures (7&8). Study of mesh independent is considered to guarantee that the results found is not relay with the mesh resolution. **Figure (7)** represents the water depth elevation and **Figure (8)** shows the velocity profiles downstream of the roughness elements for each mesh size. The results show that there is not important change happen in the level of water depth with mesh size increment after 650,000 cells mesh size. Thus 650,000 cells mesh size was considered in this study for the CFD simulation and comparable mesh geometry was adopted for all models. Not all the Ghazal (9) experiments were used—only those where a cubic artificial bed roughness and discharge of 0.01 m³/s were used in the comparison. The simulations were continued until the free surface height at the upstream and downstream of the artificial bed roughness stopped varying. **Figures (9&10)** for free surface and velocity profiles downstream the roughness elements show that the flow reached a steady value after 65 seconds from the beginning of the simulation.
Figure (7): Free surface profiles show mesh independency.

Figure (8): Velocity profiles downstream bed roughness show mesh independency.
**Figure (9):** Free surface profiles show time independency.

**Figure (10):** Velocity profiles downstream of bed roughness show time independency.
5. Results

Results of the determined water height and velocity profiles downstream of the bed roughness elements for all three configurations with flow rate 0.01 m³/s are given below for FLUENT. A correlation of the outcomes with the estimations from the exploratory investigation is additionally given. To see the degree to which the previous measure looked at Figure (11) predicted the manning values using velocity profiles with and without roughness elements by the CFD simulation. For those models apply the k-ω SST model, the CFD outcomes and experimental data (see table 1) concurred well overall. Figure (12) shows the velocity profiles downstream the bed roughness element for different configurations. The CFD simulations predict same results of many previous experimental works (11), which show that the velocity profiles decrease with increase the bed roughness elements. Figure (13) shows the water depth downstream the bed roughness zone for the three configurations (1, 2, and 3). The results agreed well the experimental outcomes which illustrate the when the density of the artificial cubic roughness elements increase the level of water depth downstream will be decrease.
Figure (11): Velocity profiles downstream of bed roughness show the Manning coefficient (n)
Figure (12): Velocity profiles downstream of bed roughness for different configuration of bed roughness elements.
Figure (13): The variation of the water level with the configurations (1, 2 and 3) where cubic roughness elements increased water level decreased
In this paper the effect of angle of attack in river with cubic bed roughness have been investigated, configuration 2 has been chosen as a case study for this investigation. Figure (14) shows the velocity profile downstream of the cubic bed roughness elements for Config-2 for different angles of attack varied from 0 – 60 degree, the results show that the velocity profile vary with respect of the angle of attack. Figure (15) shows the effect of variation of angles of attack on the flow velocity close to the bed (0.005m above the riverbed). It can be clearly seen the that the maximum velocity magnitudes and its location changed with changing the angle of attack. Figure (16) presents the bed shear stress for all cases of angles of attack. This figure shows that the bed shear stress downstream of the bed roughness elements varied with variation of angle of attack. For both Figures (15 and 16) the area of the maximum velocity magnitude close to the bed and the bed shear stresses downstream of the bed roughness elements varies with respect to angles of attack.

**Figure (14):** Velocity profiles downstream of bed roughness in Config-2 for different angles of attack.
Figure (15): Velocity contour lines at distance of 0.005m above the riverbed for Config-2 bed roughness and with different angles of attack.
Figure (16): Bed shear stress contour lines at distance of 0.005m above the riverbed for Config.-2 bed roughness and with different angles of attack.
6. Conclusions and Future Work

The first part of this research is to discuss the accuracy of CFD with Volume of Fluid (VOF) method to predict the characteristics of flow above cubic bed roughness elements with different configurations. By using previous experimental works as an approval material, it has been demonstrated that it is conceivable to get a CFD model that having the option to get results with worthy degrees of precision. The second part of this investigation is to find the effect of these bed roughness element when it’s located at a non-straight river on the flow characteristics downstream the bed roughness zone. The results show that the velocity profile at the centerline of the river downstream of the bed roughness zone decrease with increase the angle of attack, but the area of the maximum velocity magnitude close to the bed and the bed shear stresses downstream of the bed roughness elements varies with respect to angles of attack.

7. References
1. Saeed SM. Hydraulic resistance of prismatic roughness elements in steep channels: University of Mosul, Iraq; 1990.
2. Graf W, Blanckaert K, editors. Flow around bends in rivers. 2th international conference, new trends in water and environmental engineering for safety and life: eco-compatible solutions for aquatic environments, Capri (Italy); 2002.
3. Pradhan A, Kumar Khatua K, Sankalp S. Variation of Velocity Distribution in Rough Meandering Channels. Advances in Civil Engineering. 2018;2018.
4. Ghazal AM. Manning's coefficient for geometric roughness elements: University of Baghdad; 2015.
5. Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics. 1981;39(1):201-25.
6. Sarker M, Rhodes D. Calculation of free-surface profile over a rectangular broad-crested weir. Flow measurement and Instrumentation. 2004;15(4):215-9.
7. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA journal. 1994;32(8):1598-605.
8. Hargreaves D, Morvan H, Wright N. Validation of the volume of fluid method for free surface calculation: the broad-crested weir. Engineering Applications of Computational Fluid Mechanics. 2007;1(2):136-46.
9. Ghazal AM. Manning's coefficient for geometric roughness elements: University of Baghdad; 2015.
10. Matsson J. An Introduction to ANSYS Fluent SDC Publications; 2019.
11. Samer TI. Velocity Distribution in a River Cross Section with Variable Roughness: University of Baghdad; 2017.