Validation of Computational Fluid Dynamics Approach of Lateral Velocity Profile Due to Curvature Effect on Floodplain Levee of Two-stage Meandering Channel

Shriya K. Rathor1 · Abinash Mohanta1 · K. C. Patra2

Received: 25 May 2022 / Accepted: 27 August 2022 / Published online: 16 September 2022 © The Author(s), under exclusive licence to Springer Nature B.V. 2022

Abstract
Rivers play an integral part in the sustenance of life on earth. The river or natural channel’s sinuous nature causes erosion of its outer wall and silt deposition on its inner wall. These changes in the terrain make it vital to design the meandering compound channel. Levees, both straight and meandering, can be constructed on both sides of the channel to guard the surrounding lands. For doubly meandering levee alignment, the momentum transfer between the main channel and the adjoining floodplains affects the velocity distribution across the channel cross-section. So, hydraulic engineers must investigate this complex flow phenomenon in the case of a doubly meandering compound channel. This study investigates the flow in meandering compound channels using numerical and physical modelling, both of which are important in understanding non-uniform flow and its behaviour. As a result, a research is presented on the sharing of velocity over the cross section of the main channel and the floodplains of meandering compound channels. The geometry of meandering compound channels, having straight as well as sinuous floodplain levees, is chosen for this study. Unlike the experimental models, numerical hydraulic models are remarkably economical. So, in this research, the numerical hydraulic model is adopted to determine the different flow characteristics of compound meandering channels. In this paper, a complete three-dimensional and two-phase Computational Fluid Dynamics (CFD) model of the meandering compound channels is analyzed making use of the RNG K-ε turbulence model as well as Volume of Fluid (VOF) methodology. A comparison study between numerical results and experimental results are represented, considering the two different experimental data of compound meandering channel having the main channel meandered with sinuosity (ratio of channel length to the down valley length) of 1.37 and were flanked by straight levees.

Highlights
- Experimental investigation on meandering compound channel varying the sinuosity of floodplain levee.
- Comparative numerical computational fluid dynamics (CFD) analysis considering RNG K-ε turbulence model.
- Performed Error Analysis for checking compatibility of Numerical approach.

Extended author information available on the last page of the article
floodplains as well as meandering floodplains having sinuosity 1.06 on both sides. By error analysis, it is observed that the numerical results agree with that of the experimental ones.

**Keywords** Boundary shear stress distribution · Computational fluid dynamics · Meandering compound channel · Sinuosity · Turbulence model · Velocity distribution

### 1 Introduction

Scientists and engineers have always been intrigued by rivers. It is the primary source of water for household, agricultural, industrial, transport, and recreational purposes. However, designing and organising these systems demands a thorough understanding of flow and sediment transport mechanics. Channels of rivers do not maintain a straight path for long periods. In various hydraulic systems, such as irrigation networks, bridges, flumes, aqueducts, power tunnels, and siphons, flow separation in open channel expansion has been identified as one of the primary challenges. The study of non-prismatic open channels is gaining interest due to conditions like super-elevation, secondary flows, and their tendency to redistribute mean velocity, permitting the boundary shear stress, bank erosion and shifting, flow separation, and bed migration in mobile boundary channels. In majority of the cases, the flows are subcritical. The flow in compound meandering channel sections can create a continual change in kinetic energy, which can be converted to pressure energy. Due to the shifting flow conditions in this operation, energy losses occur in the channel compression. Furthermore, the presence of an unfavourable pressure gradient creates flow separation as a result of the flow’s failure to adhere to the boundaries, as well as losses of head due to the formation of eddies. Controlling the flow separation is necessary to reduce bed and bank erosion. Transition walls were once used to prevent flow separation. As a result, it is beneficial in hydraulic engineering to examine open channel expansion structures in order to evaluate velocity distribution, boundary conditions, shear distribution, flow separation management, and suitable hydraulic structural designs. The three-dimensional nature of the flow makes any modelling approach for estimating flow parameters complicated. Because of the varied hydraulic conditions in compound channels, the flow structures in the main channel and floodplains differ. The bed shear and velocity are critical in determining the anisotropy and behaviour of this complex fluid flow. The velocity profile in the sections varies due to the differences in flow structure, resulting in differing velocity distributions. The shear force distribution at the channel beds and walls allows for a thorough understanding of bed load transfer and channel pattern alterations. Computational fluid dynamics (CFD) is a computerized numerical analysis tool. Researchers have found an increasing interest in using CFD-based simulation in several disciplines of engineering as numerical hydraulic models can greatly cut expenses associated with experimental models. After applying specified boundary conditions, the basic principle of CFD is to investigate fluid flow in detail by solving a system of nonlinear governing equations across the region of interest. A numerical investigation of a non-prismatic compound channel flow with converging floodplains has been undertaken. The work will aid in simulating the various flow variables in complicated flow geometry.

Various studies have investigated the lateral velocity profile, sharing of depth averaged velocity as well as shear force, transference of momentum, and discharge calculation in meandering compound channels (Toebes and Sooky 1967; Kar 1977; Das 1984; Kiely 1989; Willetts and Hardwick 1993; Patra and Kar 2000; Khatua 2007; Mohanty 2013; Pradhan and Khatua 2017; Mohanta 2019; Mohanta et al. 2022). Also numerous investigations were carried out to examine the flow pattern at channel confluence considering various geometrical
parameters by using numerical methods (Biron et al. 2004; Czernuszenko and Rylov 2002; Ramamurthy et al. 2006; Goudarzizadeh et al. 2011; Goudarzizadeh et al. 2010; Shabayek et al. 2002; Schindfessel et al. 2015; Karami et al. 2017; Schindfessel et al. 2017). Kiely (1989) and McKeogh and Kiely (1989) investigated on compound channels and concluded that the high turbulence intensities were seen in floodplain area of compound meandering channel and not in the straight compound channel in longitudinal direction. Depth-averaged velocity profile and shear stress spreading in rough channels were studied and an equation for depth averaged velocity in a rough channel which include dip phenomenon by showing impact of water surface profile was formulated by Yang (2010). Sarma et al. (2000) developed binary version or summed up form of velocity distribution, in which logarithmic law was applied for the inner section and parabolic law for the outer section to define velocity distribution in open channels. Wilkerson and McGahan (2005) modeled depth-averaged velocity in simple trapezoidal channel. Knight et al. (2007) modified Shiono and Knight Method (SKM) (Shiono and Knight 1988) considering secondary flow effect to examine the lateral distributions of depth-averaged velocity and boundary shear stress in straight prismatic channels. Afzal et al. (2007) studied the roughness factor and turbulent effect in roughed pipe and channels through power law of velocity profile. Lane et al. (2004) performed CFD analysis by changing bed topography and roughness on gravel-bed river and also evaluated wall roughness height in individual cells based on cell porosity and technological capabilities. Carney et al. (2006) investigated natural channels with substantial bed roughness to offer a strategy to mitigate the coarse bed utility through CFD analysis. A comparative study of Reynolds Averaged Navier–Stokes (RANS) and Double Averaged Navier–Stokes (DANS) turbulent models were performed by Rameshwaran et al. (2011) for fine as well as coarse gravel bed, widely spaced pebble cluster bed in meandering river. The findings of Rameshwaran et al. (2011) provided a wide applicability of the RANS model to simulate depth averaged velocity and turbulent kinetic energy vertical profile. Gandhi et al. (2016) investigated on variation of two dimensional velocity profiles considering various channel bed slope, upstream bend and a convergence/divergence of channel width. The negatively buoyant flow in a diverging channel with sloping bottom was analyzed by Ahmad et al. (1999) using k–ε turbulence model. Singh et al. (2018) carried out CFD simulations of depth averaged velocity and boundary shear stress for in-bank flow on open channel considering gravel bed and showed the compatibility of numerical results with experimental. Mohanta et al. (2015) and Mohanta and patra (2018) studied non-uniform flow and their effect on velocity and boundary shear distribution through experimental as well as Large Eddy Simulation (LES) turbulence model in numerical CFD approach on converging compound channel. Singh et al. (2020) studied flow interaction between the main channel and floodplain of asymmetric compound channel by using Reynold’s stress model through computational (CFD) analysis. Singh et al. (2022) introduced modified interactive channel division method for discharge estimation in asymmetric compound channel and also Rahimi et al. (2022) studied the momentum transfer in steady and unsteady flow.

The semi-logarithmic relationship between velocity and distance from the wall is characterised as the ‘law of wall,’ which exists outside the laminar zone. This is important in the context of numerical modelling since the choice of standard and scalable wall functions, as well as the near wall treatment or wall function. These wall functions allow the transition of simulation results from no slip near wall condition to turbulent flow away from the wall (Carney et al. 2006).

Using ANSYS-FLUENT, a computational fluid dynamics (CFD) modelling tool, an effort has been undertaken to analyse the velocity profiles for two separate sections of compound meandering channels with straight and meandering floodplain levee on both sides.
of compound channel. The CFD model was used to investigate the impacts of a meandering floodplain levee as well as variations in velocity profiles in both horizontal and vertical directions. The velocity profile obtained by numerical simulation was compared to the actual measurement carried out by testing in the same channel utilising Preston tube.

2 Experimental Setup

At the hydraulics-engineering laboratory of the department of civil engineering, National Institute of Technology Rourkela (NITR) India, the reported experimental investigations were carried out in two-stage meandering channels with sinuosity (ratio of length of meandering channel to the down valley length) 1.37 for main channel and two different floodplain levee arrangement cases such as straight floodplain levee (sinuosity = 1) on both sides of floodplain and meandering floodplain levee of sinuosity 1.06. The second channel is also termed as doubly meandering compound channel as both main channel and floodplain levee are meandering in nature. Inside a cemented flume of 10 m length, 1.7 m width and 0.25 m depth, these meandering channels with floodplain were cast out of perspex sheets. Table 1 summarises the experimental setup for these compound channels. The specifications of the experimental setup of the meandering compound channels are provided in Fig. 1 for a clear understanding.

The bottom width \( b \) and bank full depth \( h \) of the main channel were kept fixed at 0.28 m and 0.12 m for all meandering compound channels. Perspex sheets with a roughness of 0.01 were used to create the channel surface.

The channel bed slope is maintained as 0.001 which is observed by calculating the difference in flow depths at upstream and downstream of the channel while the tailgate is closed. A movable bridge was installed across the flume, with the ability to traverse

| Sl. No | Parameters | NITR Type I* | NITR Type III*** |
|--------|------------|--------------|-----------------|
| 1      | Type of Bed surface | Smooth | Smooth |
| 2      | Bed Slope of the Channel \( (S_o) \) | 0.001 | 0.001 |
| 3      | Angle of Arc of main channel \( (\phi_m) \) | 60° | 60° |
| 4      | Angle of Arc of floodplain \( (\phi_f) \) | 0 | 30° |
| 5      | Sinuosity of the main channel \( (s_{mc}) \) | 1.37 | 1.37 |
| 6      | Sinuosity of the Floodplain \( (s_{fp}) \) | 1 | 1.06 |
| 7      | Wavelength of the channel \( (\lambda) \) | 2.23 m | 2.23 m |
| 8      | Width of the main Channel \( (b) \) | 0.28 m | 0.28 m |
| 9      | Total width of the channel \( (B) \) | 1.67 m | 1.35 m |
| 10     | Bankfull Depth of main channel \( (h) \) | 0.12 m | 0.12 m |
| 11     | Width of outer Floodplain \( (b_o) \) | 0.25 m | 0.25 m |
| 12     | Width of inner floodplain \( (b_i) \) | 1.14 m | 0.82 m |
| 13     | Meander Belt Width \( (B_{MW}) \) | 1.17 m | 1.17 m |

*Meandering compound channel of 60° sinuosity of primary channel with straight floodplain levee; ***Doubly meandering compound channel 60° sinuosity of primary channel with 30° sinuosity of floodplain levee
over the flume in both directions for obtaining measurements at various points throughout the channel portion. Water was pumped from the subsurface sump to an overhead tank, from which water flows through the flume. The water from the flume is collected in a measuring tank at the downstream end of the flume and directed to the sump, creating a complete recirculating water supply system. For the experimental process of open channel, it was taken care that the flow within the flume takes place only due to gravity. Figure 2 shows a schematic representation of the compound channel’s experimental configurations. Longitudinal velocity, depth averaged velocity, and boundary shear stress are measured at the bend apex (section A) and cross over (section C). Figure 3 shows photographs of NITR Type I and Type III meandering compound channels.

Fig. 1 Geometric Configuration of Meandering Compound Channel a National Institute of Technology Rourkela (NITR) Type I channel; b NITR Type III channel from Mohanta (2019)

Fig. 2 Plan Geometry of the Experimental Setup
Experimental Procedure

For measuring the depth of flow in the compound meandering channels, a point gauge with a resolution of 0.1 mm having a vernier scale with a least count of 0.1 mm was used. The entire measurement instrument was attached to a movable bridge which was operated manually. The flow was kept laminar and uniform to provide a smooth execution of the experimental process and to measure the flow depth. A continuous uniform run of water flow for 4 to 5 h was necessary to obtain a stable and precise flow depth, after which flow depth was measured using the point gauge. The stilling chambers and baffle wall structure at the entrance via which water from the overhead tank travels away to the channel was also built to lessen turbulence of the flow. The point velocities of flow in the compound channels were estimated using a Preston tube with an outer diameter of 4.77 mm connected with manometer for measuring pressure difference at various specified positions within the channel cross-section. All of the readings were taken at the predetermined grid positions at the bend apex and geometric crossover of meandering compound channels as shown in Fig. 4 throughout the channel section laterally and vertically for various flow depths.

The velocity is measured laterally across the cross-sections using pitot tube. The differential pressure ($\Delta P$) was then calculated from the vertical manometer readings using the following expression

\[ \Delta P = \rho g \Delta H \]  

where $\Delta P$ is the difference between the stagnation pressure and the wall static pressure, $\Delta H$ the difference between the two head readings from the dynamic and static pressure head, $g$ the acceleration due to gravity and $\rho$ the density of water. Here the tube coefficient is taken as unity and the error due to turbulence is considered negligible. The experimental result of mean velocity and discharge are given in Table 2 with non-dimensional channel parameters.
Description of Numerical Simulation

ANSYS Fluent, a Computational Fluid Dynamics simulation tool based on the three-dimensional form of Navier–Stokes equations, is implemented for the comparative study with experimental results. The turbulent flow of compound meandering channels are primarily caused by the channel shape or geometry and gravity force. Turbulence in a compound channel is highly complicated, and the flow structure involved generates uncertainty in flow variable prediction. Large shear layers formed by the velocity difference between the main channel and the floodplain generalise turbulent formations, especially in meandering compound channels. Vortices form in both the longitudinal and vertical directions in this huge shear layer region.

Volume of fluid (VOF) and RNG $K$-$\varepsilon$ was employed as principal equations in free-surface modelling of compound meandering channels. Both are discretized in both space and time, which necessitates transient simulation. The pressure implicit splitting of operators (PISO) is used in fluent simulation to solve the connection between pressure and velocity field in this work. The transient problem is calculated using the pressure implicit splitting of operators (PISO) iterative solution method, which helps to converge the issues faster. In the simulation PRESTO spatial discretization of pressure is used to discretize second order upwind momentum equation which generally gives better results near the boundary region. The second order implicit transient formulation was also used for the simulation. The numerical solution is converged when the discretized transport equation’s residuals reach a value of $1 \times 10^{-6}$ or when the solution does not change with additional iterations. The difference in mass flow rates at the velocity input and pressure outlet was monitored to be less than 0.01 percent in the final

Fig. 4  Configuration of boundary segments for the experimental sections (Bendapex and crossover) of the meandering compound channels considering a NITR Type I; b NITR Type III

4 Description of Numerical Simulation

ANSYS Fluent, a Computational Fluid Dynamics simulation tool based on the three-dimensional form of Navier–Stokes equations, is implemented for the comparative study with experimental results. The turbulent flow of compound meandering channels are primarily caused by the channel shape or geometry and gravity force. Turbulence in a compound channel is highly complicated, and the flow structure involved generates uncertainty in flow variable prediction. Large shear layers formed by the velocity difference between the main channel and the floodplain generalise turbulent formations, especially in meandering compound channels. Vortices form in both the longitudinal and vertical directions in this huge shear layer region.

Volume of fluid (VOF) and RNG $K$-$\varepsilon$ was employed as principal equations in free-surface modelling of compound meandering channels. Both are discretized in both space and time, which necessitates transient simulation. The pressure implicit splitting of operators (PISO) is used in fluent simulation to solve the connection between pressure and velocity field in this work. The transient problem is calculated using the pressure implicit splitting of operators (PISO) iterative solution method, which helps to converge the issues faster. In the simulation PRESTO spatial discretization of pressure is used to discretize second order upwind momentum equation which generally gives better results near the boundary region. The second order implicit transient formulation was also used for the simulation. The numerical solution is converged when the discretized transport equation’s residuals reach a value of $1 \times 10^{-6}$ or when the solution does not change with additional iterations. The difference in mass flow rates at the velocity input and pressure outlet was monitored to be less than 0.01 percent in the final
Table 2 Details of Experimental Flow Conditions

| Meandering Compound Channel | Depth \((H) \text{ (m)}\) | Relative Depth \(\beta = (H-h)/H\) | Total Width \((B) \text{ (m)}\) | Width Ration \((\alpha = B/b)\) | Meander Belt Width Ratio \((\omega = B_{MW}/B)\) | Velocity \((V) \text{ (m}^2/\text{s}^2)\) | Discharge \((Q) \text{ (m}^3/\text{s})\) |
|-----------------------------|-----------------|----------------|-----------------|----------------|----------------|----------------|----------------|
| NITR Type-I (Bend-apex)     | 0.17            | 0.29           | 1.67            | 5.96           | 0.70           | 0.45           | 0.05           |
|                             | 0.195           | 0.38           | 1.67            | 5.96           | 0.70           | 0.46           | 0.07           |
| NITR Type-I (crossover)     | 0.17            | 0.29           | 2.10            | 4.96           | 0.84           | 0.51           | 0.05           |
|                             | 0.195           | 0.38           | 2.10            | 4.96           | 0.84           | 0.53           | 0.07           |
| NITR Type-III (bend-apex)   | 0.17            | 0.29           | 1.35            | 4.82           | 0.87           | 0.27           | 0.03           |
|                             | 0.195           | 0.38           | 1.35            | 4.82           | 0.87           | 0.37           | 0.05           |
| NITR Type-III (crossover)   | 0.17            | 0.29           | 1.36            | 4.86           | 0.86           | 0.27           | 0.03           |
|                             | 0.195           | 0.38           | 1.36            | 4.86           | 0.86           | 0.37           | 0.05           |
solution for simulations with an unsteady solver. Furthermore, a number of extra time steps were added to the final solution to ensure that the flow field is stable.

Present study emphasizes the multiphase flow analysis for free surface modelling with two Eulerian phases where air as primary and water as secondary phase. The secondary phase is not distributed within the first phase, but rather there is an interface between the phases, which must be tracked while solving a momentum equation for each phase. The investigation was carried out using the open channel volume of fluid (VOF) method and the implicit body force volume fraction formulation. The solution of the continuity equation for the volume fraction of one (or more) of the phases creates the interfaces between the phases. This equation has the following form for the qth phase:

$$\frac{\partial \alpha_q}{\partial t} + \nabla \cdot (\varrho \cdot \alpha_q) = 0$$

where $\alpha_q$ is the qth phase’s volume fraction. The volume fractions of all phases in each control volume add up to unity. Each cell can have one of the following three conditions:
- If $\alpha_q = 0$, the cell is empty.
- If $\alpha_q = 1$, the cell is full.
- If $0 < \alpha_q < 1$, the cell contains the interface between the $q$th phase and one or more other phases.

The average attributes of each phase are computed in each cell based on the volume fraction of each phase. The VOF method was created to track the incompressible viscous flow’s moving free surface.

Figure 5 shows the cross sectional geometry of two-stage meandering channels generated by the CFD software ANSYS (Fluent). To solve the governing equations, ANSYS (Fluent) implemented the finite element approach, which divides the region of interest into a finite number of cells (the mesh or grid). The complete domain is discretized using the finite volume discretization approach, and the meandering compound channels are discretized (meshed) as illustrated in Fig. 6.

### 4.1 Setting of Boundary Conditions and Numerical Methods

Several boundary conditions were taken into account when performing numerical modelling and simulation. It was necessary to consider the flow parameters at the inlet, outlet, wall, channel bottom, and free surface. A mean velocity is delivered over the whole inlet plane to initiate the flow, upon which velocity variations are applied. In this scenario, the intake was treated as a velocity inlet, and the average velocity calculated from experimental average values was presented in Table 2 so that the results could be compared. The inlet flow velocity is in the positive X-direction.

The outlet was assumed to be a pressure outlet. No-slip wall condition was used to simulate the channel walls and the channel bottom. No-slip boundary condition states that the fluid close to the wall assumes the velocity at the wall, which is zero, i.e. $U = V = W = 0$. The symmetry boundary condition was employed for the top free surface. This condition implies that scalar flux does not flow across the border. As a result, neither convective nor diffusive flow is present across the top surface. Normal velocities are set to zero and all other attributes outside the domain are equal to their values at the nearest node just inside the domain when this condition is implemented.

The channel geometries were all built flat to ease slope adjustments and indicate the pressure gradient. Gravity and channel slope effects are implemented via resolving the gravity vector in x, y and z components as provided in Eq. (3).
where $\theta$ is the angle between the bed surface and the horizontal axis, and $\tan \theta$ is the channel slope. The $x$ component determines the direction in which water flows through the channel, whereas the $z$ component determines the hydrostatic pressure on the channel bed. Generally in case of open channel, the "$z" component of the gravity vector ($-\rho g \cos \theta$) is found to be the cause of the solver’s convergence in the simulation. For the numerical simulation of flow in meandering compound channels, a spatial discretization technique was used with pressure as presto, momentum as quick, volume fraction as compressive, and transient formulation as constrained 2nd order implicit.

5 Results and Discussion

5.1 Measurement of Velocity Profile

As the channel is made up of four consecutive meander waves, the measuring reach was set at 4.46 m from the compound channel’s start to assure a fully developed and stable flow. Bend-apex and crossover are the two parts addressed in this study. Local velocities were measured across the full cross section, laterally and vertically at the $0.2h$, $0.4h$, \( \frac{h}{2} \), and $h$ points.
0.6 \( h \), 0.8 \( h \), and 0.9 \( h \) levels, where "\( h \)" represents the height of water for a specific segment of the channel. Local velocities are recorded at nodes in various sections of the longitudinal direction of flow, at varied depths and widths.

### 5.2 Lateral Distribution of Longitudinal Velocity Profile

Figures 7, 8, 9 and 10 show the variation of longitudinal velocity profiles at bend apex and crossover for different overbank flow conditions with two depth ratios (Dr) or relative depths (\( \beta \)) (Dr = \( \beta \) = 0.3 and 0.38) of compound meandering channel with straight floodplain levee (NITR Type I) and compound meandering channel with meandering floodplain levee or doubly meandering channel (NITR Type III).

The graphs shown above depict a clear comparison between the experimental and numerical values on NITR Type I and NITR Type III channels those have straight and meandering floodplain levees for two relative depth of flow i.e. \( \beta = 0.3 \) and 0.38. From the results of longitudinal velocity profile it is observed that velocity at the boundary of the channel is less than anywhere else, the reason for this being the boundary layer conditions. On altering the geometry of the floodplain and the height of the main channel, velocity profile changes. This can be observed by the varying values of longitudinal velocities mentioned in the graph. The variation of velocity is more in the floodplain.
than in the main channel. All these numerical results agree with the experimental results. Results of lateral distribution of longitudinal velocity profile at bend-apex and crossover of meandering compound channels have shown the compatibility of CFD approach in open channel flow study.

5.3 Lateral Distribution of Depth-averaged Velocity

In all compound channel studies, depth averaged velocity ($V_d$) is a critical parameter that must be evaluated with sufficient precision to identify its distribution across the flow section for various relative depths ($\beta$) in order to more correctly estimate discharge. At several experimental portions of the compound meandering channels, the depth-averaged velocity distribution in a cross section was measured. Equation (4) is used to define the depth-averaged velocity ($V_d$).
The point velocity measured at a flow depth of 0.4 \( h \) from the channel bottom or 0.6 \( h \) from the water surface can be approximated by depth averaged velocity at a given section of a channel (Chaudhry 2008; Rantz 1982). As a result, depth averaged velocities for meandering compound channels were recorded at varied overbank flow depths.

After the CFD simulation, an attempt has been made for testing the strength of numerical results (Fig. 11). A comparison is made between the numerical and experimental values of depth averaged velocity at bend-apex and crossover of meandering compound channels, as shown in Fig. 12a, b, respectively. The experimental results of depth averaged velocity of the NITR Type I and NITR Type III meandering compound channels at bend-apex and crossover for different depth ratios (\( \text{Dr}=0.3 \) and 0.38) are shown in Figs. 12 and 13, respectively. The correlation coefficient (\( R^2 \)) between experimental and numerical values for different cross section and different relative depth of meandering compound channels are also shown in Fig. 11.

\[
V_d = \frac{1}{h} \int_0^h Vdy
\]  

(4)

Fig. 8 Lateral Distribution of Longitudinal Velocity profile at Bend-apex for various relative depths for overbank flow of NITR Type III channel at a Outer Floodplain; b Main Channel; c Inner Floodplain
A value of $R^2$ close to 1 indicates a good agreement between the numerical and experimental value of depth averaged velocity.

**5.4 Error Analysis**

Testing the performance of numerical modeling is important where the comparison between experimental ($V_{Exp.}$) and numerical ($V_{Num.}$) velocity result should give the minimum error between the values. To assess the compatibility of numerical models, the percentage error (Eq. (5)) between experimental ($V_{Exp.}$) and numerical ($V_{Num.}$) findings of depth averaged velocity at bend-apex and crossover of the two compound meandering channels is calculated. Figures 14 and 15 show the percentage error of depth averaged velocity of meandering compound channels at the bend apex and crossover, respectively.
Performance of CFD analysis is demonstrated through their standard statistical error calculation such as root mean square error (RMSE), mean absolute error (MAE), mean absolute percentage error (MAPE), scattered index (SI), and efficiency (E). These parameters are evaluated by the Eqs. (6) through (10) (Mohanta 2019). The experimental value (Exp) and numerical value (Num) are the mean values of experimental and numerical results, respectively, and $n$ is the total number of observations in error analysis. Performances evaluation in terms of RMSE, MAE, MAPE, SI, E for the numerical approach are given in Table 3.

\[
\text{Error(\%)} = \left( \frac{V_{\text{Exp.}} - V_{\text{Num.}}}{V_{\text{Exp.}}} \right) \times 100
\]

Fig. 10 Lateral Distribution of Longitudinal Velocity profile at Crossover for various relative depths for overbank flow of NITR Type III channel at a Outer Floodplain; b Main Channel; c Inner Floodplain.
Fig. 11 Comparison between numerical and experimental values of depth averaged velocity of compound meandering channels considering a bend-apex; b crossover

\[
RMSE = \sqrt{\frac{1}{n} \sum_{i=1}^{n} (\text{Exp}_i - \text{Num}_i)^2}
\]  \hspace{1cm} (6)

\[
MAE = \frac{1}{n} \sum_{i=1}^{n} |\text{Exp}_i - \text{Num}_i|
\]  \hspace{1cm} (7)

Fig. 12 Lateral distribution of Depth averaged velocity for overbank flow at bend-apex of Channels a NITR Type-I; b NITR Type III
Fig. 13  Lateral distribution of Depth averaged velocity for overbank flow at crossover of various channels a NITR Type I; b NITR Type III

\[
MAPE = \frac{1}{n} \sum_{i=1}^{n} \left( \frac{|Exp_i - Num_i|}{Exp_i} \times 100 \right)
\]  

\[
SI = \sqrt{\frac{\sum_{i=1}^{n} \left( \left( Exp_i - \overline{Exp} \right) - \left( Num_i - \overline{Num} \right) \right)^2}{\sum_{i=1}^{n} Exp_i^2}}
\]  

\[
E = 1 - \frac{\sum_{i=1}^{n} \left( Exp_i - Num_i \right)^2}{\sum_{i=1}^{n} \left( Exp_i - \overline{Exp} \right)^2}
\]

Fig. 14  Percentage of error of depth-averaged velocity for different position laterally throughout the bend-apex considering various meandering compound channels a NITR Type-I; b NITR Type-III
MAE measures the closeness of numerical and experimental values, where the RMSE indicates the deviation of numerical value from experimental value. MAE and RMSE have units similar to the unit of depth averaged velocity (m/s). Lower values of MAE and RMSE depict a better performance of CFD modeling. Efficiency, $E = 1$ or nearer to 1 corresponds to a perfect match between the experimental and numerical results, which can be ranged between $-∞$ to 1. The SI is a normalized measure of error; the lower SI value indicates the better performance of CFD. The better results regarding error analysis show the acceptability of the numerical method for the practical application to study flow analysis in meandering compound channels.

### 5.5 Distribution of Velocity Contours

Velocity counters can be used to illustrate the variation of longitudinal velocities across channel sections. Figures 16, 17, 18 and 19 illustrate velocity counters at bend-apex and crossover sections for NITR Type I and NITR Type III meandering compound channels.

| Table 3 | Performance metrics for numerical modeling |
|---|---|---|---|---|---|---|---|---|
| Channel | Cross section | Relative Depth (Dr) | MAE | RMSE | MAPE | E | SI |
| NITR Type–I | Bend-apex | 0.3 | 0.0178 | 0.0225 | 0.049 | 0.33 | 0.049 |
| NITR Type–I | 0.38 | 0.0208 | 0.0241 | 0.052 | 0.81 | 0.053 |
| NITR Type–III | 0.3 | 0.0213 | 0.0251 | 0.057 | 0.94 | 0.051 |
| NITR Type–III | 0.38 | 0.0250 | 0.0290 | 0.051 | 0.97 | 0.041 |
| NITR Type–I | Crossover | 0.3 | 0.0219 | 0.0238 | 0.062 | 0.69 | 0.075 |
| NITR Type–I | 0.38 | 0.0153 | 0.0190 | 0.039 | 0.87 | 0.045 |
| NITR Type–III | 0.3 | 0.0152 | 0.0196 | 0.038 | 0.97 | 0.053 |
| NITR Type–III | 0.38 | 0.0240 | 0.0280 | 0.047 | 0.97 | 0.060 |
with a flow depth of 0.195 m (depth ratio, $Dr = 0.38$). Similarly, for a flow depth of 0.17 m, i.e. depth ratio, $Dr = 0.3$, velocity contours of NITR Type I and NITR Type III meandering compound channels at bend-apex and crossover sections are depicted in Figs. 20, 21, 22 and 23.

The contours of longitudinal velocities (Figs. 16, 17, 18, 19, 20, 21, 22 and 23) for meandering compound channels have shown that the lowest velocity contour lines are observed nearer to the outer floodplain wall at bend-apex. The velocity fields reveal growing tendencies towards the channel’s inner side. The highest velocity fields are also shown nearer to the inner floodplain wall as it is nearer to the convex meander wall at the bend apex. Near the free surface and in the inner flood plain, stream wise velocity reaches its maximum magnitude. The width of the right floodplain (inner floodplain) rapidly diminishes as one moves from the bend apex to the crossover, and the velocity filament tends to retard to the main channel, i.e. the floodplain flows enter the main channel and mixing occurs at the interface. The strongest velocity field advances toward the outer wall of the main channel and the outer floodplain region as we move from bend-apex to cross-over. Because the bend apex section is closer to the convex side of the meander cross-over. Because the bend apex section is closer to the convex side of the meander wall, the highest velocity field occurs near the inner floodplain wall.
Fig. 17 Velocity Contours at Bendapex for the depth ratio (Dr) as 0.38 of NITR Type-III channel at (a) Experimental; (b) Numerical

Fig. 18 Velocity Contours at Crossover for the depth ratio (Dr) as 0.38 of NITR Type-I channel at (a) Experimental; (b) Numerical
Fig. 19 Velocity Contours at Crossover for the depth ratio (Dr) as 0.38 of NITR Typ-III channel at a Experimental; b Numerical

Fig. 20 Velocity Contours at Bendapex for the depth ratio (Dr) as 0.3 of NITR Type-I channel at a Experimental; b Numerical
Fig. 21 Velocity Contours at Crossover for the depth ratio (Dr) as 0.3 of NITR Type-I channel at a Experimental; b Numerical

Fig. 22 Velocity Contours at Bendapex for the depth ratio (Dr) as 0.3 of NITR Type-III channel at a Experimental; b Numerical
Conclusions

In the present study, the experimental and numerical modelling of the flow pattern at meandering compound channels with straight as well as meandering floodplain levee has been carried out. Findings of the work are as follows:

1. Mesh refinement studies and experiments have effectively proven comprehensive three-dimensional modelling of free surface flow in meandering compound channel and doubly meandering compound channel.

2. The numerical analysis for longitudinal velocities and depth-averaged velocity are compared to the corresponding values of experimental results for two different sections, i.e. bend-apex and crossover of the meandering compound channel and doubly meandering compound channel, for two different relative depths of flow, and it is concluded that longitudinal velocity variation in the main channel region is less than that in the flood plain.

3. Both experimental and numerical results show that the velocity is nearly constant after the solid boundary in the main channel region, whereas there is rapid variation in the flood plain region, with maximum velocity occurring just below the free surface of the converging compound channel. This is because the interaction between water surfaces...
of top layer and air forms a boundary layer, which affects the velocity profile near to the free surface.

4. Based on the velocity contour, it is concluded that the findings of the CFD analysis and the experimental data are in good accord.

5. The velocity contour depicts that the maximum velocity is found in the compound channel’s central region and it gradually decreases towards the boundary at cross section and the maximum velocity is found near to inner wall of main channel as well as floodplain at bend-apex.

6. The numerical simulation shows that, when compared to other turbulence models, the RNG $K-\varepsilon$ turbulence model, which considers near wall modelling, has good agreement with experimental data for a compound channel.

Acknowledgements The authors acknowledge the support from the Department of Civil Engineering, National Institute of Technology Rourkela, India, to conduct the experiments.

Author Contributions A.M. did experimental work. S.K.R performed CFD simulations. S.K.R. and A.M. analyzed and validated the results, prepared the graphs, and wrote the manuscript. K.C.P. reviewed the results and provided feedback on this paper. All Authors reviewed the manuscript.

Data Availability All data, models, and code generated or used during the study appear in the submitted article. In detail data can be found from the relevant literatures.

Declarations

Ethics Approval Not applicable.

Consent to Participate Not applicable.

Consent to Publish The authors declare their consent to publication of the manuscript in “Water Resources Management” journal.

Competing Interests Not applicable.

References

Afzal N, Seena A, Bushra A (2007) Power law velocity profile in fully developed turbulent pipe and channel flows. J Hydraul Eng 133(9):1080–1086
Ahmad Z, Kothyari UC, Ranga Raju KG (1999) Longitudinal dispersion in open channels. ISH J Hydraul Eng 5(2):1–21
Biron PM, Ramamurthy AS, Han S (2004) Three-dimensional numerical modeling of mixing at river confluences. J Hydraul Eng 130(3):243–253
Carney SK, Bledsoe BP, Gessler D (2006) Representing the bed roughness of coarse-grained streams in computational fluid dynamics. Earth Surf Proc Land 31(6):736–749
Chaudhry MH (2008) Two-dimensional flow. Open-Channel Flow, Springer US, Boston, MA, 407–452
Czernuszenko W, Rylov A (2002) Modeling of three-dimensional velocity field in open channel flows. J Hydraul Res 40(2):135–143
Das AK (1984) A study of river flood plain interaction and boundary shear stress distribution in a meander channel with one sided flood plain. IIT, Kharagpur
Gandhi BK, Verma HK, Abraham B (2016) Mathematical modeling and simulation of flow velocity profile for rectangular open channels. ISH J Hydraul Eng 22(2):193–203
Goudarzizadeh R, Jahromi SHM, Hedayat N (2010) Simulation of 3D flow using numerical model at open-channel confluences. World Acad Sci Eng Technol 4(11):1437–1442
Goudarzizadeh R, Mousavi JSH, Hedayat N, Naghshineh MH (2011) Numerical investigation of the flow separation zone and quality of the bed shear stress distribution at open-channel confluences. Proc Word Acad Sci Eng Technol, Dubai, UAE, 1231–1235
Kar SK (1977) A study of distribution of boundary shear in meander channel with and without floodplain and river floodplain interaction. Indian Institute of Technology Kharagpur Kharagpur, India
Karami H, Farzin S, Sadrabadi MT, Moazeni H (2017) Simulation of flow pattern at rectangular lateral intake with different dike and submerged vane scenarios. Water Sci Eng 10(3):246–255
Khatua KK (2007) Interaction of flow and estimation of discharge in two stage meandering compound channels. Natl Inst Technol Rourkela
Kiely GK (1989) An experimental study of overbank flow in straight and meandering compound channels. NUI, at Department of Civil Engineering, UCC
Knight DW, Omran M, Tang X (2007) Modeling depth-averaged velocity and boundary shear in trapezoidal channels with secondary flows. J Hydraul Eng 133(1):39–47
Lane SN, Hardy RJ, Keylock CJ (2004) Coherent flow structures in shallow gravel-bed rivers: an experimental. Shallow Flows: Research Presented at the International Symposium on Shallow Flows, Delft, Netherlands, 2003, 259
McKeogh EJ, Kiely GK (1989) Experimental study of the mechanisms of flood flow in meandering channels. Proceedings of the 23rd IAHR Congress, 491–498
Mohanta A (2019) Modelling of overbank flow in two-stage meandering channels. PhD., National Institute of Technology Rourkela, Rourkela, India
Mohanta A, Khatua KK, Patra KC (2015) Flow modeling in symmetrically narrowing flood plains. Aquat Procedia 4:826–833
Mohanta A, Patra KC (2018) LES modeling with experimental validation of a compound channel having converging floodplain. J Inst Eng (India) Ser A 99(3):519–537
Mohanta A, Pradhan A, Patra KC (2022) Determination of discharge distribution in meandering compound channels using machine learning techniques. J Irrig Drain Eng 148(1):04021063
Mohanty PK (2013) Flow analysis of compound channels with wide flood plains Prabir. Natl Inst Technol Rourkela
Patra KC, Kar SK (2000) Flow interaction of meandering river with floodplains. J Hydraul Eng 126(8):593–604
Pradhan A, Khatua KK (2017) Assessment of roughness coefficient for meandering compound channels. KSCE J Civ Eng 22(5):2010–2022
Rahimi H, Yuan S, Tang X, Lu C, Singh P, Dehrashid FA (2022) Study on conveyance coefficient influenced by momentum exchange under steady and unsteady flows in compound open channels. Water Resour Manag 1–21
Ramamurthy AS, Qu J, Zhai C (2006) 3D simulation of combining flows in 90 rectangular closed conduits. J Hydraul Eng 132(2):214–218
Rameshwaran P, Naden PS, Lawless M (2011) Flow modelling in gravel-bed rivers: rethinking the bottom boundary condition. Earth Surf Proc Land 36(10):1350–1366
Rantz SE (1982) Measurement and computation of streamflow. US Department of the Interior, Geological Survey, Washington DC
Sarma KV, Prasad BVR, Sarma AK (2000) Detailed study of binary law for open channels. J Hydraul Eng 126(3):210–214
Schindfessel L, Creelle S, De Mulder T (2015) Flow patterns in an open channel confluence with increasingly dominant tributary inflow. Water 7(9):4724–4751
Schindfessel L, Creelle S, De Mulder T (2017) How different cross-sectional shapes influence the separation zone of an open-channel confluence. J Hydraul Eng 143(9):04017036
Shabayek S, Steffler P, Hicks FE (2002) Dynamic model for subcritical combining flows in channel junctions. J Hydraul Eng 128(9):821–828
Shiono K, Knight DW (1988) Refined modelling and turbulence measurements. Proc Int Symp IAHR, Tokyo, Japan, 26–28
Singh PK, Banerjee S, Naik B, Kumar A, Khatua KK (2018) Lateral distribution of depth average velocity & boundary shear stress in a gravel bed open channel flow. ISH J Hydraul Eng 27(1):23–37
Singh PK, Tang X, Rahimi H (2020) A computational study of interaction of main channel and floodplain: open channel flows. J Appl Math Phys 8(11):2526–2539
Singh PK, Tang X, Rahimi H (2022) Linear-scale models for discharge estimation: Asymmetric compound open channel flows. Proc Inst Civil Eng-Water Manag 149–161
Toebes GH, Sooky AA (1967) Hydraulics of meandering rivers with flood plains. J Water Harbors Div 93(2):213–236
Wilkerson GV, McGahan IL (2005) Depth-averaged velocity distribution in straight trapezoidal channels. J Hydraul Eng 131(6):509–512
Willetts BB, Hardwick RI (1993) Stage dependency for overbank flow in meandering channels. Proc Inst Civil Eng-Water Marit Energy 101(1):45–54
Yang S-Q (2010) Depth-averaged shear stress and velocity in open-channel flows. J Hydraul Eng 136(11):952–958

**Publisher’s Note** Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

Springer Nature or its licensor holds exclusive rights to this article under a publishing agreement with the author(s) or other rightsholder(s); author self-archiving of the accepted manuscript version of this article is solely governed by the terms of such publishing agreement and applicable law.

**Authors and Affiliations**

Shriya K. Rathor¹ · Abinash Mohanta¹ · K. C. Patra²

¹ Abinash Mohanta
abinash.mohanta@vit.ac.in

Shriya K. Rathor
rathor.k.shriya@gmail.com

K. C. Patra
kcpatra@nitrkl.ac.in

¹ School of Mechanical Engineering, Vellore Institute of Technology, Vellore, Tamil-Nadu 632014, India

² Dept. of Civil Engineering, National Institute of Technology Rourkela, Rourkela 769008, India