

Coupled three-dimensional hydrodynamics and sediment deposition mechanisms at a bifurcation in an open-channel water conveyance system

Meichen Zhao

x20231010015@stu.ncwu.edu.cn

NCWU: North China University of Water Resources and Electric Power

Jian Chen

North China University of Water Resources and Electric Power

Mingjia Dong

NCWU: North China University of Water Resources and Electric Power

Hai Xue

NCWU: North China University of Water Resources and Electric Power

Lina Yin

Henan Provincial Department of Natural Resources

Zhiheng Li

NCWU: North China University of Water Resources and Electric Power

Yaowei Li

NCWU: North China University of Water Resources and Electric Power

Yangyang Tian

NCWU: North China University of Water Resources and Electric Power

Research Article

Keywords: blind-ended channel reach, diversion channel, circulation, sediment deposition

Posted Date: September 24th, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-10969786/v1>

License:  This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Coupled three-dimensional hydrodynamics and sediment deposition mechanisms at a bifurcation in an open-channel water conveyance system

Meichen Zhao¹, Jian Chen^{1*}, Mingjia Dong¹, Hai Xue¹, Lina Yin², Zhiheng Li¹, Yaowei Li¹, Yangyang Tian¹

1. School of Water Conservancy, North China University of Water Resources and Electric Power, Zhengzhou, 450046, Henan, China

2. Institute of Natural Resources Monitoring and Land Consolidation, Department of Natural Resources of Henan Province, Henan 450016, China

(Received month, date, year, Revised month, date, year, Accepted month, date, year, Published online month, date, year)
 ©China Ship Scientific Research Center 2026

Abstract: In long-distance water diversion projects, channel bifurcation zones are prone to developing a three-dimensional flow structure characterized by a “blind-channel reach,” which consequently induces sediment deposition in the inlet region. Taking a diversion channel of the Middle Route of the South-to-North Water Diversion Project as a case study, a three-dimensional numerical model is established to systematically investigate the flow characteristics in the inlet region as well as the transport and deposition behavior of suspended particles. The results indicate that flow separation of the main current occurs in the inlet area. Under the combined effects of shear stress and pressure gradient, a three-dimensional circulation system is formed through the coupling of horizontal and vertical vortical structures. In this process, flow separation provides the spatial condition for circulation development, shear stress acts as the driving mechanism, and the water level difference maintains the stability of the circulation pattern. Based on Lagrangian particle tracking of sediments with different densities, the transport pathways and deposition processes of particles in the inlet region are further elucidated. A coupled sedimentation mechanism is proposed, characterized by “main flow deflection – circulation-induced retention and re-entrainment – particle settling,” which clarifies the intrinsic hydrodynamic mechanism governing sediment accumulation in channel bifurcation zones.

Key words: blind-ended channel reach; diversion channel; circulation; sediment deposition

0. Introduction

In water transfer projects, the downstream end of a channel is often in a closed condition, resulting in a channel configuration characterized by an “open inlet and closed end,” within which backflow phenomena similar to those observed in blind-ended channel reaches may develop in the inlet region[1]. Experimental studies on flow partitioning in diversion channels have primarily been conducted using the momentum equation[2][3], energy equation[4], and dimensional analysis[5][6][7]. The diversion width can be determined based on streamline patterns[8][9]. Different diversion angles can lead to significant variations in pressure distribution, turbulence intensity, and energy loss in the bifurcation region[10][11][12].

Computational fluid dynamics (CFD) provides a framework in which physical models are established and the Navier–Stokes (N–S) equations are solved numerically to obtain solutions under specified boundary conditions[13]. Both the Reynolds-averaged approach and the RNG k – ϵ turbulence model have been demonstrated to be effective for numerical simulations of open-channel flow division[14][15][16].

The flow structure at bifurcation regions exhibits strong three-dimensional characteristics, this structure is similar to that observed in curved channel flows[17]. Such complex hydrodynamic features lead to substantial differences between sediment diversion ratios and water discharge ratios in bifurcated channels[18]. In practical water transfer systems, sediment concentrations in branch channels are typically higher than those in the main channel, and severe sediment deposition is often observed along the bed of branch channels[19]. Such deposition processes may reduce the hydraulic efficiency of water conveyance systems and increase maintenance

Project supported by the National Natural Science Foundation of China. (Grant No. U22A20237)

Biography: Meichen Zhao (2001-), Male, M.S.,
 E-mail: X20231010015@stu.ncwu.edu.cn

Corresponding author: Jian Chen,
 E-mail: chenjian@ncwu.edu.cn

requirements. To address this issue, scholars have discovered that guide vanes can weaken and disrupt the lateral circulation at the branch outlet, making them an effective measure to prevent sediment particles from entering the branch channel[20][21][22].

Previous studies have mainly focused on conventional sediment particles; however, in open-channel water transfer systems, impurities are dominated by fine-grained sediments and organic matter. Their transport and deposition behaviors at bifurcation zones remain insufficiently understood. In addition, existing research has rarely addressed the coupled effects of planar circulation, vertical circulation, and sediment deposition processes. The fact that water flow structure can enhance particle residence time, causing particles to undergo multiple transport, resuspension, and sedimentation processes is overlooked. And the sediment accumulation in water transfer channels cannot be fully explained only by local velocity reduction or particle settling velocity. Therefore, a three-dimensional hydrodynamic perspective is essential for revealing the interaction mechanism between complex flow structures and sediment accumulation.

This study aims to investigate the influence mechanisms of sediment deposition from the perspective of three-dimensional hydrodynamic structures. Based on three-dimensional numerical simulations, the RNG k- ϵ turbulence model and a Lagrangian particle tracking model are employed to analyze the three-dimensional hydrodynamic characteristics of diversion channels and their inlet regions under complex flow conditions, and to elucidate the deposition mechanisms of impurity particles from the viewpoint of flow structure.

1. Methodology and Framework Overview

1.1 Renormalization Group turbulence model

In nature, most fluid motions exhibit turbulent characteristics. Turbulence models are therefore commonly employed to simulate energy dissipation induced by vortical structures in viscous flows[23]. The RNG (Renormalization Group) turbulence model is theoretically based on the renormalization group approach[24]. By modifying the statistical characteristics of eddies across different scales and introducing additional terms into the governing equations, the model achieves improved numerical accuracy under complex flow conditions. In particular, it provides more stable and reliable results when simulating flow separation, strong shear flows, and localized recirculation phenomena.

The transport equations for turbulent kinetic energy (k) and its dissipation rate (ϵ) are given as follows:

Turbulent kinetic energy equation:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\alpha_k (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] G_k - \rho \quad (1)$$

Dissipation rate equation:

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\alpha_s (\mu + \mu_t) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon}^* \frac{\epsilon}{k} G_k - C_{2\epsilon} \frac{\rho \epsilon^2}{k} \quad (2)$$

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}; G_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j};$$

$$C_{1\epsilon}^* = C_{1\epsilon} - \frac{\eta \left(1 - \frac{\eta}{\eta_0} \right)}{1 + \beta \eta^3} \quad (3)$$

where: μ_t denotes the turbulent viscosity coefficient; $C_\mu = 0.0845$; G_k represents the production term of turbulent kinetic energy induced by mean velocity gradients; $C_{1\epsilon} = 1.42$ [25], $C_{2\epsilon} = 1.68$; α_k and α_s are the effective Prandtl numbers, both equal to 1.39.

1.2 The Lagrangian particle model

The Lagrangian particle tracking model treats particles as independent discrete phases and tracks their individual trajectories within the flow field, thereby enabling the simulation of particle transport, settling, and deposition processes. Because the Lagrangian framework directly follows the time evolution of particle trajectories[26], it provides a more intuitive description of transport, dispersion, and settling processes in fluids, and can explicitly represent particle paths as well as their interactions with the surrounding flow field.

The particle motion equation is given as:

$$\frac{d u_p}{d t} = -\frac{1}{\rho_p} \nabla P + g + \beta (u - u_p) |u - u_p| \cdot \frac{\rho}{\rho_p} + \frac{m_{added}}{m_p} \left(\frac{d u}{d t} - \frac{d u_p}{d t} \right) \quad (4)$$

where: u_p and ρ_p denote the particle velocity and particle density, respectively; g represents gravitational acceleration acting on the particle; u and P are the fluid velocity and pressure, respectively; β is the drag-related coefficient; m_p is the particle mass; and m_{added} denotes the added mass of the surrounding fluid. For spherical particles, the added mass is assumed to be equal to one-half of the mass of

the displaced fluid.

1.3 Initial conditions

First, a preliminary simulation was conducted for the area encompassing the discharge channel and the 190-meter stretch of the upstream main channel to characterize the natural settling process of particles released from the inlet boundary. Regarding boundary conditions: the upstream channel boundary was defined as a pressure inlet, with inflow conditions controlled by specified flow rate and water level; particles were released continuously and uniformly at the inlet cross-section at a rate of 1,000 particles per second; the downstream boundary at the channel end was defined as a pressure outlet; the channel bottom and side walls were set as no-slip solid boundaries; the top boundary was defined as an atmospheric pressure boundary; the initial water level throughout the computational domain was set to 7 m; the fluid medium was defined as an incompressible fluid at 20°C, with gravitational acceleration set to -9.81 m/s^2 ; and the channel roughness coefficient was set to 0.015.

1.4 Operating conditions

Table 1

Summary of simulation cases and particle properties

case	Flow rate (m^3/s)	Water depth (m)	Particle density (kg/m^3)	Particle diameter (mm)
case 1	164	7.09	1200	0.4
case 2	190	7.5	1200	0.4
case 3	200	7.38	1200	0.4
case 4	164	7.09	2650	0.15
case 5	190	7.5	2650	0.15
case 6	200	7.38	2650	0.15

1.5 Numerical model reliability assessment

To evaluate the reliability of the numerical model, a mesh sensitivity analysis was conducted using different spatial resolutions (Tab. 2). The variations in velocity distribution, circulation scale, and pressure field under different mesh configurations were compared. As the mesh resolution increased, the simulated results gradually approached a stable condition, indicating satisfactory mesh independence. After comprehensively considering error margins, computational efficiency, and the formation of circulation patterns, a uniform cubic mesh with dimensions of $d_x = d_y = d_z = 0.5 \text{ m}$ was selected, resulting in a total of over 2.4 million mesh cells across the computational domain.

Table 2

Mesh independence analysis

Mesh size(m)	Cells	Maximum velocity difference
1	3.1×10^5	4.75%

In natural water bodies and water conveyance channels, the settling behavior of sediment and dust particles is significantly influenced by particle size and density[27]. Generally, particles with larger sizes and higher densities are more strongly affected by gravity, resulting in higher settling velocities and a greater tendency to deposit in regions of reduced flow velocity. In contrast, smaller and lighter particles are more likely to remain suspended and be transported with the flow, exhibiting stronger dispersion and retention under turbulent conditions.

Accordingly, two representative particle types are defined to characterize different impurity constituents. Table 1 presents the parameter settings under different flow conditions. Particle Type 1 represents suspended particles in the inlet region of the diversion channel, while Particle Type 2 represents sediment particles entering the channel from the natural environment. In this study, all particles are assumed to be spherical, and inter-particle collisions are neglected.

0.8	6.1×10^5	2.79%
0.6	1.4×10^6	1.31%
0.5	2.4×10^6	0.82%
0.4	4.8×10^6	0.16%

Since this study mainly focuses on flow separation and circulation development in the bifurcation region, model evaluation was performed not only based on velocity convergence but also on the variation of circulation characteristics. The location and intensity of vortex cores exhibited limited changes after further mesh refinement, demonstrating that the numerical model can effectively capture the complex three-dimensional flow structures within the diversion region.

2. Experimental Results and Discussion

2.1 Planar Circulation

As shown in Fig. 1, to overcome the limitation of conventional velocity vector plots, which are mainly

qualitative, the Q -criterion is introduced to quantitatively identify the planar circulation structure in the inlet region. The results indicate a distinct region with $Q > 0$ in the inlet zone, representing the dominance of local rotational motion and the formation of a stable planar circulation structure. This high- Q region is primarily distributed near the lower boundary of the inlet, and the Q -value gradually decays along the channel direction, indicating that the

circulation intensity weakens downstream from the circulation core. The flow velocity near the lower boundary is generally higher than that near the upper boundary; however, within the circulation region, the overall velocity level remains significantly lower than that in the main channel. The horizontal and vertical axes of the circulation extend along the channel cross-sectional direction and the channel centerline direction, respectively.

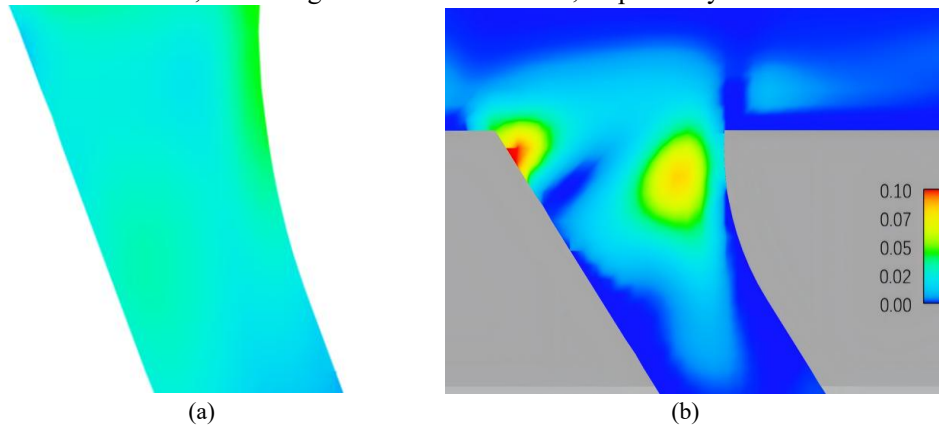


Fig. 1 Vector plot and contour plot at 4.2m water depth

From a hydrodynamic perspective (Fig. 2), momentum exchange occurs between the main flow and the water body in the inlet region. This exchange is manifested as the continuous action of shear stress exerted by the main flow on the inlet water body. Under sustained shear stress, the fluid near the interface is driven to move downstream along with the main flow. In addition, the presence of a downstream revetment at the inlet (Fig. 2(b)) leads to a rise in water level near the lower boundary, generating a local water-level difference relative to the surrounding flow. This results in a positive pressure gradient. Under this pressure difference, the water near the lower boundary flows toward both the interior of the diversion channel, where the surrounding water level is lower, and toward the upper boundary, forming a

gravity-compensated back flow characteristic of blind-ended channel reaches, thereby sustaining a quasi-steady circulation.

In summary, in the inlet region, the main flow separates from the interface due to abrupt geometric changes and an adverse pressure gradient, providing spatial conditions for circulation development. Meanwhile, shear stress induced by velocity gradients promotes momentum exchange, serving as the driving force of the circulation. In addition, the rise in water level induced by the revetment leads to a local hydraulic head difference, which drives return flow into the diversion channel under pressure gradient effects, thereby maintaining the stability and persistence of the circulation structure

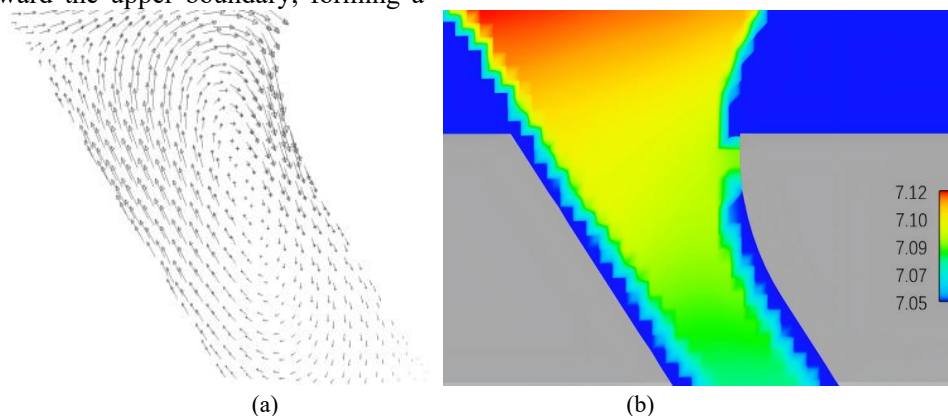


Fig. 2 Vector plot and contour plot at 4.2m water depth

As the flow from the main canal passes the entrance to the diversion channel, a sudden change in

boundary conditions causes the main flow to separate from the interface at the entrance, rapidly pushing the

water away from that interface and into the diversion channel. As shown in Figure 3, the water entering the diversion channel can be subdivided into three parts based on flow paths and directions: water in Zone 1 moves in a direction approximately parallel to the canal's central axis immediately after entering the diversion channel, without reaching the downstream canal wall; water in Zone 2 changes direction upon reaching the bank slope and flows back into the main canal; and water in Zone 3 changes direction upon reaching the bank slope and flows into the diversion channel.

Consequently, the pressure difference resulting from the change in boundary conditions causes the water in the main flow region to separate from the interface, thereby triggering the flow zonation at the entrance. The results indicate distinct vertical stratification in the flow field: water in the bottom

2.2 Vertical Circulation

As shown in Fig. 4, distinct closed circulation structures are observed at all representative cross-sections (i.e., the lower wall, channel center line, and upper wall). The core regions of these circulations correspond to pronounced $Q > 0$ zones, indicating the dominance of rotational effects and the formation of stable vortex cores. The maximum Q values at the three sections are approximately 0.069 s^{-2} , 0.063 s^{-2} , and 0.082 s^{-2} , respectively, while the corresponding

layer enters the diversion channel due to the deflection of the main flow; water in the middle layer exhibits a recirculating flow structure; and water in the upper layer flows back toward the main canal's flow region in the opposite direction

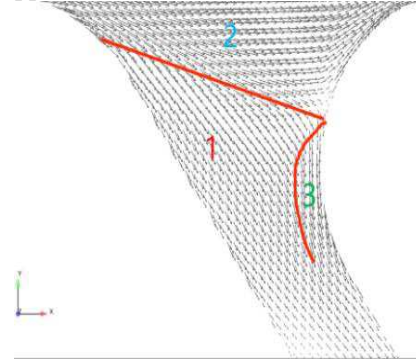


Fig.3 Vector plots of flow velocity at different water depths

maximum shear stress values are 0.40 Pa, 0.38 Pa, and 0.51 Pa. The formation of vertical-plane circulation is primarily driven by shear stress induced by vertical velocity gradients, and the circulation intensity is closely related to local shear stress. Both quantities exhibit a gradual variation trend from the lower wall toward the upper wall. This phenomenon indicates that the vertical-plane circulation exhibits pronounced spatial non-uniformity.

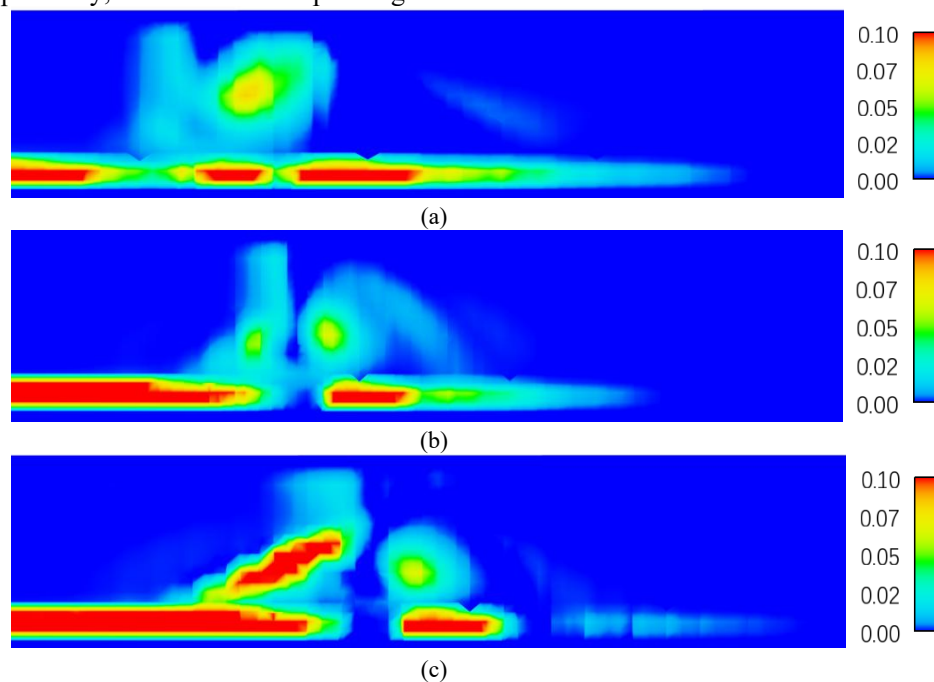


Fig. 4 Longitudinal profile velocity distribution and Q-criterion vortex structure contour plot

A portion of the water along the bank slope is also diverted away from the main flow and into the channel (Fig. 5). The flow entering the entrance zone

from the channel bed generates shear stress due to the velocity gradient relative to the existing water in that zone; this alters the flow direction of the surface water,

and the three water components collectively form a vertical circulation structure. In terms of spatial distribution, the centers of the vertical circulation vary across different cross-sections: from the upper wall to the lower wall, the centers are located at depths of approximately 4.54 m, 4.09 m, and 3.09 m, respectively. Velocity vector plots reveal that water at the bottom moves along the channel bed into the discharge channel, while water in the upper layer flows back in the opposite direction toward the main channel flow, establishing a classic "inflow at the bottom, outflow at the top" vertical circulation pattern.

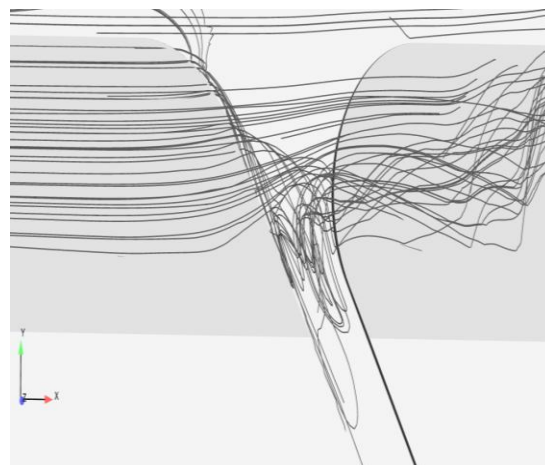


Fig.5 Flow diagram of entrance area

3. Analysis of particle sedimentation

3.1 Qualitative analysis of particle sedimentation

As shown in Figure. 6, after entering the inlet region, impurity particles gradually deposit along the diversion channel, exhibiting an overall banded spatial distribution. Meanwhile, influenced by the vertical-plane circulation structure, a portion of the particles is transported upward into the upper water layer and re-enters the main flow, demonstrating a clear re-transport phenomenon. However, significant differences are observed in the spatial distribution patterns of different types of impurity particles within the inlet region and the diversion channel.

Particles with a density of 2650 kg/m^3 tend to settle toward the channel bed after entering the inlet region. Their deposition zones are mainly concentrated in the inlet region and the upstream section of the diversion channel, with deposition preferentially occurring near the upper wall, while

relatively less deposition is observed near the lower wall (Fig. 7). In contrast, particles with a density of 1200 kg/m^3 remain in a suspended state after entering the diversion channel and are further transported downstream, exhibiting a more dispersed spatial distribution.

From a mechanistic perspective, these differences are primarily governed by the competition between gravitational settling and hydrodynamic disturbance intensity. The low-density particles (1200 kg/m^3) exhibit weak settling capacity, strong responsiveness to the flow field, and are significantly influenced by three-dimensional circulation and turbulence, resulting in stronger spatial dispersion. In contrast, high-density particles (2650 kg/m^3) have higher settling velocities and are more likely to decouple from the flow field. After entering the inlet region, they rapidly leave the main flow under gravity dominance and undergo deposition.

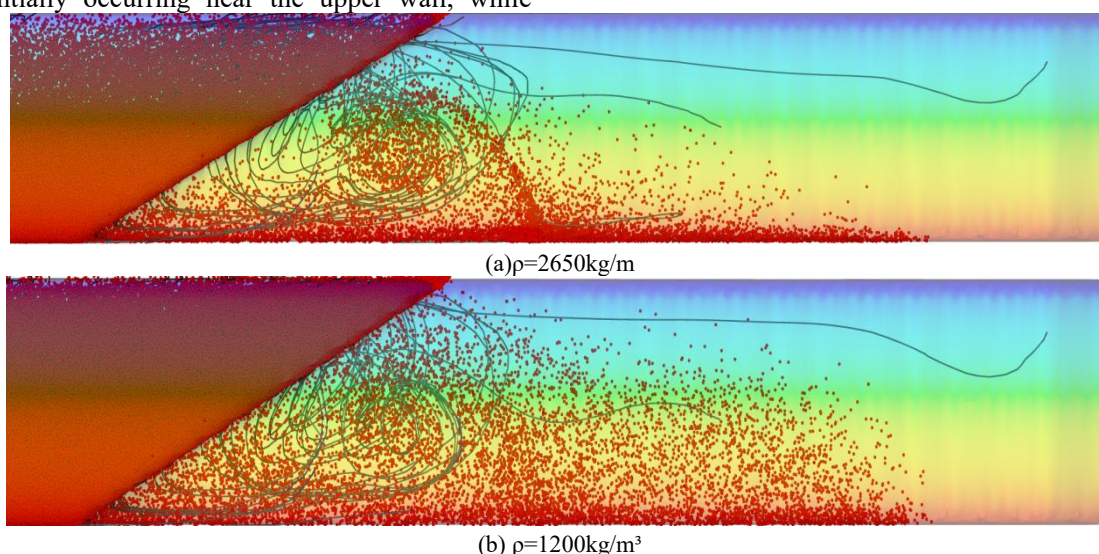


Fig. 6 Side view of the distribution of different particles

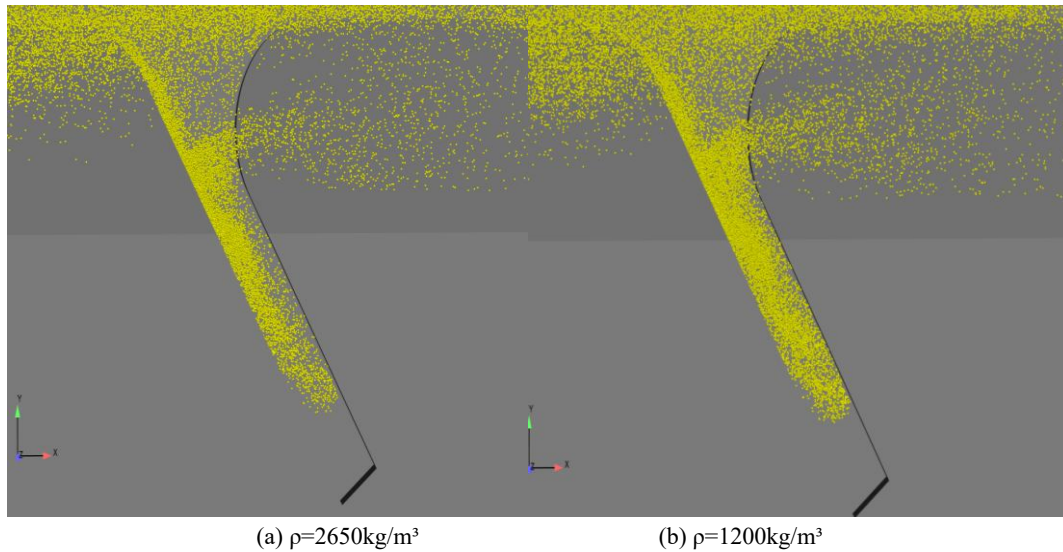


Fig. 7 Top view of the distribution of different particles

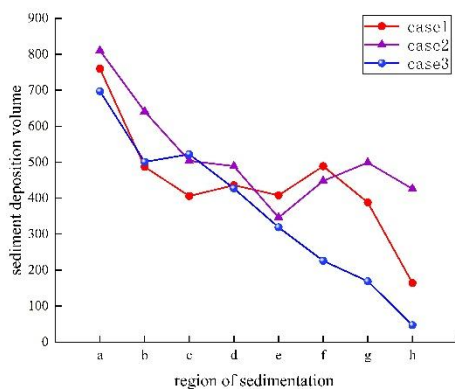
3.2 Quantitative analysis of particle sedimentation

By arranging monitoring cross-sections along the channel and obtaining the sediment quantity in a given region through subtracting particle counts between adjacent sections, the spatial distribution of deposition can be quantified. In this study, sections labeled a, b, and c correspond to the inlet region, while section d and downstream sections represent the interior of the diversion channel. Figure 8 presents the simulated results of particle deposition within the channel.

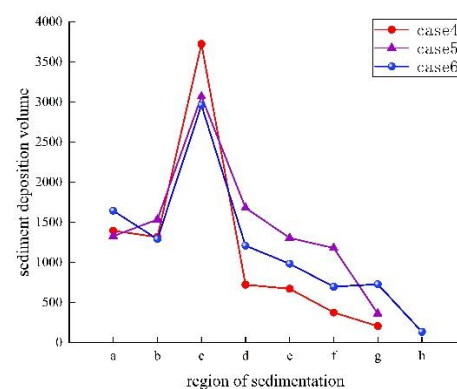
For particles with a density of 1200 kg/m³, their enhanced suspension capacity enables them to be more readily transported downstream by the main flow, resulting in a relative reduction in the number of

particles entering the diversion channel from the channel bed and exhibiting a more dispersed distribution pattern. For particles with a density of 2650 kg/m³, the increase in Reynolds number enhances the transport capacity of near-bed flow, leading to a reduction in deposition within the inlet region, while the deposition extent and amount within the diversion channel increase correspondingly.

These results indicate that particle transport and deposition are jointly controlled by the combined effects of momentum and gravitational settling, and that different particle types exhibit significantly different responses to variations in the flow field.



(a) $\rho=1200\text{kg/m}^3$



(b) $\rho=2650\text{kg/m}^3$

Figure 8: Accumulated sediment/deposition quantity of different particles in the drainage channel

3.2 Discussion

For water transfer channels characterized by blind-ended configurations, particle transport processes are not governed solely by particle settling properties. Instead, due to the pronounced flow separation and recirculation structures developed near

the inlet region, particle migration involves a complex sequence of transport, retention, and re-transport processes jointly regulated by local three-dimensional hydrodynamic structures and intrinsic particle characteristics.

Flow separation acts as the initial driving

mechanism controlling particle entry into the bifurcation region and subsequent deposition. When the main flow approaches the inlet zone, the abrupt variation in channel geometry induces streamline deviation and flow separation under the combined effects of boundary constraints and pressure gradients. Consequently, particles originally transported along the main flow pathway undergo lateral displacement and enter the bifurcation region. This process alters the original transport trajectories of particles and provides the material source for subsequent accumulation.

After entering the inlet region, particle migration is further controlled by the three-dimensional circulation structures. The planar circulation establishes a local recirculating transport pathway, which weakens the downstream advective transport capacity and increases particle residence time within the inlet zone. Meanwhile, the vertical circulation modifies particle exchange between different water layers, enabling particles to migrate between the near-bed and upper water regions. This vertical exchange process increases the complexity and instability of particle trajectories, allowing certain particles to undergo repeated cycles of suspension, migration, and deposition. Compared with conventional sedimentation analyses that mainly consider longitudinal velocity variations, the three-dimensional circulation framework provides a more reasonable explanation for the long-term retention and localized accumulation of particles in bifurcation inlet regions.

The final deposition behavior of particles is determined by the competition between hydrodynamic disturbance and gravitational settling. Under high shear stress and strong turbulent fluctuations, hydrodynamic forces are sufficient to maintain particles in suspension and sustain their circulation-driven transport. However, when particles enter low-velocity recirculation zones, the reduction in local shear stress and turbulent diffusion capacity promotes the transition from suspended transport to deposition. For particles with a density of 2650 kg/m^3 , the relatively high settling velocity enables them to gradually decouple from the surrounding flow field, resulting in preferential deposition within the inlet region and the upstream section of the diversion channel. In contrast, particles with a density of 1200 kg/m^3 exhibit weaker settling capability and stronger dependence on local three-dimensional flow structures, leading to enhanced transport capacity and a more dispersed spatial distribution. These results indicate

that particle density does not directly determine deposition outcomes; rather, it regulates deposition patterns by modifying the response of particles to complex hydrodynamic structures.

3. Conclusions

This study employs numerical simulation to systematically investigate the three-dimensional flow characteristics and particle sedimentation patterns at the diversion junction of an open-channel water conveyance system. It reveals the complex flow structure at the junction and the mechanisms by which this structure influences particle migration and deposition. The results indicate that the flow structure in the junction area exhibits distinct three-dimensional characteristics. Regarding horizontal circulation, the ambient water at the junction moves forward under the influence of shear stress, causing water piling up against the bank slope; driven by the water level difference, the flow enters the diversion channel, thereby establishing horizontal circulation. Regarding vertical circulation, the flow enters the junction area from the bottom, and its direction shifts due to shear stress, creating a vertical circulation pattern. A stable three-dimensional circulation structure emerges at the junction through the combined effects of main flow separation, shear stress generated by velocity gradients, and the water level difference. Specifically, main flow separation provides the spatial conditions for circulation formation, shear stress supplies the driving force, and gravity compensation sustains the circulation.

Funding

This work was supported by the National Natural Science Foundation of China. (Grant No. U22A20237)

Conflict of interest

All authors declare that there are no other competing interests.

Author contributions

All authors contributed to the study conception and design. Material preparation, data collection and analysis were performed by [Mingjia Dong], [Zhiheng Li] and [Jian Chen]. The first draft of the manuscript was written by [Meichen Zhao] and all authors commented on previous versions of the manuscript. All authors read and approved the final manuscript.

Data Availability

The numerical simulation data of this study are available from the first author or corresponding author

upon reasonable request.

Ethics approval and Consent to participate

Not applicable

Acknowledgment

The authors would like to thank the relevant engineering management authorities for providing valuable information related to the study area.

References

- [1] Odgaard, A. J., & Kennedy, J. F. River-bend bank protection by submerged vanes [J]. *Journal of hydraulic engineering*, 1983, 109(8), 1161-1173.
- [2] Ramamurthy, A. S., Minh Tran, D., & Carballada, L. B. Dividing flow in open channels [J]. *Journal of Hydraulic Engineering*, 1990, 116(3), 449-455.
- [3] Lama, S. K., Kuroki, M., & Hasegawa, K. Study of flow bifurcation at tic 30° open channel junction when the width ratio of branch channel to main channel is large [J]. *Proceedings of Hydraulic Engineering*, 2002, 46, 583-588.
- [4] Hager, W. H. Lateral outflow over side weirs [J]. *Journal of Hydraulic Engineering*, 1987, 113(4), 491-504.
- [5] Mignot, E., Zeng, C., Dominguez, G., Li, C. W., Rivière, N., & Bazin, P. H. Impact of topographic obstacles on the discharge distribution in open-channel bifurcations [J]. *Journal of hydrology*, 2013, 494, 10-19.
- [6] Rivière, N., Travin, G., & Perkins, R. J. Trans critical flows in three and four branch open-channel intersections [J]. *Journal of Hydraulic Engineering*, 2014, 140(4), 04014003.
- [7] Al-Zubaidy, R., & Hilo, A. N. Numerical Study of Sedimentation and Flow Pattern at the Open Channel Intake [J]. *Wasit Journal of Engineering Sciences*, 2022, 10(3), 105-122.
- [8] Ibrahim, I., Riviere, N., Lebouteiller, I., & Mignot, E. Discharge distribution in open-channel T-shape bifurcations: Effect of a reduced side branch width [J]. *Journal of Hydraulic Engineering*, 2022, 148(9), 04022015.
- [9] Moghadam, M. K., Bajestan, M. S., Sedghi, H., & Seyedian, M. (2014). An experimental and numerical study of flow patterns at a 30 degree water intake from trapezoidal and rectangular channels [J]. *Iranian Journal of Science and Technology. Transactions of Civil Engineering*, 38(C1), 85.
- [10] Alomari, N. K., Yusuf, B., Ali, T. A. M., & Ghazali, A. H. Flow in a branching open channel: a review [J]. *Pertanika Journal of Scholarly Research Reviews*, 2016, 2(2).
- [11] Alomari, N. K., Yusuf, B., Mohammad, T. A., & Ghazali, A. H. Influence of diversion angle on water and sediment flow into diversion channel [J]. *International Journal of Sediment Research*, 2020, 35(6), 600-608.
- [12] Al-Sabea, S. Z., & Hilo, A. N. The effect of lateral intake slope on sedimentation transport [J]. *Wasit Journal of Engineering Sciences*, 2024, 12(3), 152-163.
- [13] Neary, V. S., & Sotiropoulos, F. Numerical investigation of laminar flows through 90-degree diversions of rectangular cross-section [J]. *Computers & fluids*, 1996, 25(2), 95-118.
- [14] Ghostine, R., Hoteit, I., Vazquez, J., Terfous, A., Ghenaim, A., & Mose, R. Comparison between a coupled 1D-2D model and a fully 2D model for supercritical flow simulation in crossroads [J]. *Journal of Hydraulic Research*, 2015, 53(2), 274-281.
- [15] Ahadiyan, J., Chenari, S. A., Nadian, H. A., Katopodis, C., Valipour, M., Sajjadi, S. M., & Omidvarinia, M. Sustainable systems engineering by CFD modeling of lateral intake flow with flexible gate operations to improve efficient water supply [J]. *International Journal of Sediment Research*, 2024, 39(4), 629-642.
- [16] Ramamurthy, A. S., Qu, J., & Vo, D. Numerical and experimental study of dividing open-channel flows [J]. *Journal of hydraulic engineering*, 2007, 133(10), 1135-1144.
- [17] He, W., Feng, S. Y., Zhang, J., Tang, H. W., Xiao, Y., Chen, S., & Liu, C. S. Hydrodynamic characteristics and particle tracking of 90 lateral intakes at an inclined river slope [J]. *Water Science and Engineering*, 2024, 17(2), 197-208.
- [18] Alomari, N. K., Yusuf, B., Mohammad, T. A., & Ghazali, A. H. Experimental investigation of scour at a channel junctions of different diversion angles and bed width ratios [J]. *Catena*, 2018, 166, 10-20.
- [19] Pandey, A. K., & Mohapatra, P. K. 3D numerical simulations of the bed evolution at an open-channel junction in flood conditions [J]. *Journal of Irrigation and Drainage Engineering*, 2024, 150(3), 04024007.
- [20] Zhao, W., Zhang, J., He, W., Shi, L., & Chen, X. Effects of diversion wall on the hydrodynamics and withdrawal sediment of a lateral intake [J]. *Water Resources Management*, 2022, 36(3), 1057-1073.
- [21] Gungum, F., & Cardoso, A. H. Optimizing the desilting efficiency of submerged vane fields at lateral diversions [J]. *Journal of Hydraulic Engineering*, 2023, 149(1), 04022031.
- [22] Bor, A. Experimental investigation of 90 intake flow patterns with and without submerged vanes under sediment feeding conditions [J]. *Canadian Journal of Civil Engineering*, 2022, 49(3), 452-463.
- [23] Ran, D., Wang, W., & Hu, X. Three-dimensional numerical simulation of flow in trapezoidal cutthroat flumes based on FLOW-3D [J]. *Frontiers of Agricultural Science and Engineering*, 2018, 5(2), 168-176.
- [24] Yakhot, V., & Orszag, S. A. Renormalization group analysis of turbulence I Basic theory [J]. *Journal of scientific computing*, 1986, 1(1), 3-51.
- [25] Yakhot, V. S. A. S. T. B. C. G., Orszag, S. A., Thangam, S., Gatski, T. B., & Speziale, C. Development of turbulence models for shear flows by a double expansion technique [J]. *Physics of Fluids A: Fluid Dynamics*, 1992, 4(7), 1510-1520.
- [26] Baharvand, S., Ahmari, H., & Taghvaei, P. Developing a Lagrangian sediment transport model for open channel flows [J]. *International Journal of Sediment Research*, 2023, 38(2), 153-165.
- [27] Wang, H., Tai, Y., Huang, L., Yang, C., & Jing, H. Analyzing water and sediment flow patterns in circular forebays of sediment-laden rivers [J]. *Sustainability*, 2023, 15(24), 16941.

Supplementary Files

This is a list of supplementary files associated with this preprint. Click to download.

- [Right.pdf](#)