Uluslararası İleri Doğa Bilimleri ve Mühendislik Araştırmaları Dergisi
Sayı 9, S. 64-74, 8, 2025

Araştırma Makalesi

International Journal of Advanced Natural Sciences and Engineering Researches
Volume 9, pp. 64-74, 8, 2025

Research Article

https://as-proceeding.com/index.php/ijanser
ISSN:2980-0811

Influence of Ditch Width and Depth on Velocity Distribution in Irrigation Canals: A Case Study Using ANSYS Fluent

Tamjeed Attaullah*[1], Naveed Anjum[2]

[1]*Departmemt of Civil Engineering, Khawaja Fareed University of Engineering and Information Technology, Pakistan*

[2]*Departmemt of Civil Engineering, Khawaja Fareed University of Engineering and Information Technology, Pakistan*

(muhammadtamjeed26@gmail.com)

(Received: 27 July 2025, Accepted: 31 July 2025)

(3rd International Conference on Modern and Advanced Research ICMAR 2025, July 25-26, 2025)

ATIF/REFERENCE: Attaullah, T. & Anjum, N. (2025). Influence of Ditch Width and Depth on Velocity Distribution in Irrigation Canals: A Case Study Using ANSYS Fluent, *International Journal of Advanced Natural Sciences and Engineering Researches*, 9(8), 64-74.

Abstract – With an emphasis on a section of the Abbasia Canal in southern Punjab, Pakistan, this study investigates how ditch width and depth affect flow behaviour in organized irrigation canals. Three distinct ditch shapes were assessed using Computational Fluid Dynamics (CFD) modelling in ANSYS Fluent to see how they affected energy dissipation, turbulence intensity, and velocity distribution. The realizable k-ε turbulence model was utilized to simulate flow structures and energy losses, while the Volume of Fluid (VOF) approach was utilized to capture free surface flow behaviour. In order to replicate real-world field circumstances, each simulation included a perpendicular ditch that was attached to the main canal and varied in size. The findings show that ditch shape affects hydraulic efficiency by substantially changing both longitudinal and lateral velocity profiles. With its consistent velocity distribution and low turbulence, the configuration in Case 2 (3 m broad, 0.75 m deep) showed the best performance, increasing discharge capacity. However, Case 3 (6 m wide, 1.5 m deep) increased the possibility for sediment deposition and decreased flow momentum while successfully lowering peak flow velocities and wasted energy. A moderate balance between these impacts was provided by Case 1. In addition to offering practical advice for maximizing water delivery, reducing erosion, and promoting sustainable canal infrastructure in semi-arid areas, these findings highlight the significance of ditch design in irrigation canal management.

Keywords – Open Channel Flow, Ditch Geometry, ANSYS Fluent, CFD Simulation, VOF Method.

I. INTRODUCTION

The effectiveness and dependability of irrigation systems have a major impact on agricultural productivity in arid and semi-arid environments. Structured irrigation canals are essential to maintaining food security and rural development in nations like Pakistan, where agriculture is the main driver of the national economy. Among these, the Abbasia Canal is crucial for providing irrigation water to thousands of hectares of agriculture in southern Punjab. It is a significant distributary of the Rohri Canal system [1]. In addition to distributing water, the Abbasia Canal serves as a vital link for the surrounding agricultural environment, connecting to a system of field-level ditches that control water supply at the micro-irrigation level [2]. The purpose of these secondary and tertiary ditches is to guarantee that crops receive water at crucial growth stages in a timely and uniform manner. However, hydraulic inefficiencies frequently impair the effectiveness of these micro-irrigation networks. Typical difficulties include sediment deposition from upstream runoff, obstructions or scouring that change the desired hydraulic profiles, and higher flow resistance brought on by channel roughness or aquatic vegetation [3]. These problems are frequently brought on by old or insufficient channel alignment and geometry. The flow velocity, turbulence zones, and energy dissipation patterns along the canal can be significantly impacted by slight changes in ditch characteristics, such as width, depth, and slope. Crop yields and the sustainability of irrigation as a whole are impacted by these inefficiencies over time because they lead to uneven water distribution, decreased discharge efficiency, and increased maintenance costs [4]. Open channel flow has historically been modelled using empirical equations like Manning's equation, but these approaches frequently fall short of capturing the intricate dynamics brought about by submerged elements like vegetation or localized geometrical deviations [5]. The Volume of Fluid (VOF) method in conjunction with Computational Fluid Dynamics (CFD) offers a more precise and comprehensive way for simulating free-surface flow behaviour in open channels. CFD simulations provide a useful tool for design optimization by visualizing pressure changes, turbulence generation, and velocity profiles under various hydraulic and geometric situations. Notwithstanding its potential, not much research has been done to examine the precise impacts of perpendicular ditch layouts in organized irrigation canals such as Abbasia using CFD [6, 7]. Although the impact of localized ditch structures on canal hydraulics is not well measured in the literature, they are commonly included for runoff management, sediment control, or flow redirection. Evidence-based design improvements for such field situations are hampered by the knowledge gap created by the dearth of thorough simulation-based investigations. The current work fills this gap by analyzing the impact of ditch geometry on flow performance using a hydraulic simulation of the Abbasia Canal using ANSYS Fluent and the VOF method [8]. The main emphasis is on the effects of ditch width and depth variations on the energy dissipation, turbulence intensity, and velocity distribution within a specific canal segment. This study intends to generate insights that can guide better ditch design and placement methods, ultimately leading to more efficient irrigation and more successful flood management, by analyzing three different scenarios with different ditch layouts [9].

The purpose of this study is to:

- Analyse how various ditch shapes affect the canal's longitudinal and lateral velocity fields.
- Determine which setups improve or decrease hydraulic efficiency.
- In large-scale canal systems, make design suggestions that promote erosion prevention and sustainable water distribution.

The results of this study will help irrigation planners and engineers make more data-driven and simulation-supported ditch implementation decisions for current and future canal infrastructures [10]. Additionally, it will support adaptive flood mitigation plans, especially in regions where water availability varies due to climate change [11].

II. LITERATURE REVIEW

Geometric and environmental factors have a significant impact on hydraulic behaviour in open channels, especially in irrigation systems where effective water delivery depends on flow management. Flow resistance, which results from channel irregularities, vegetation, boundary roughness, and sediment deposition, is one of the main elements influencing velocity distribution and discharge capacity in such systems. Although Manning's equation and other empirical models are still frequently employed to estimate flow rates in open channels, their oversimplified assumptions frequently fall short of capturing the complexities seen in actual settings, particularly in areas with vegetation and morphological changes [2]. Both upstream and downstream velocity distributions are greatly impacted by the introduction of localized turbulence and changes in momentum exchange between flow layers caused by submerged or emergent vegetation. Studies conducted in the field and in laboratories that examine vegetated channels for ecological stability and flood management have shown this [12]. Furthermore, non-uniformities in discharge behaviour are introduced by sediment transport dynamics caused by seasonal runoff and channel slope fluctuations, which are frequently difficult for classic hydraulic models to adequately address [3, 13]. A more reliable foundation for examining such intricate hydraulic situations is provided by recent developments in computational fluid dynamics, or CFD. CFD can properly mimic free surface flows in irrigation structures by capturing velocity profiles, turbulence formation, and energy dissipation mechanisms when combined with the Volume of Fluid (VOF) approach, according to studies [14]. When secondary features like ditches, weirs, or energy dissipators are added, or when canal geometry is altered, these tools are quite helpful. Researchers have looked into using bed depressions, moats, and subsurface features in coastal and flood engineering to create hydraulic jumps and encourage energy dissipation. For example, Zaha et al. [9] showed that by lowering peak velocity and spreading shear pressures, carefully placed moat-like depressions in channel beds might decrease erosion. Similar ideas have been used to build ditches in irrigation canals, where deeper or wider sections serve as local flow stabilizers to improve sediment flushing and lessen turbulence [10, 15].

It has also been demonstrated that improving ditch design increases the canals' capacity for self-cleaning, inhibits the development of stagnant zones, and boosts hydraulic efficiency generally. Variation in ditch width and depth can have a substantial impact on energy gradients, flow continuity, and sediment transport efficiency in both lined and unlined channels [16]. Nevertheless, a considerable research gap still exists in the application of CFD-based hydraulic models to secondary ditch structures in structured canal systems, especially in semi-arid areas such as southern Punjab, Pakistan, in spite of these advancements [17]. The majority of current research ignores the effects of localized ditches and how they interact with main channel hydraulics, instead concentrating on either primary canal flows or idealized channel geometry. Additionally, few studies use ANSYS Fluent or other high-fidelity platforms to take into account the combined influence of ditch width, depth, and surrounding channel characteristics under realistic flow regimes [18]. By integrating ANSYS Fluent with the VOF model to simulate flow behaviour in the Abbasia Canal, this work seeks to close this gap. The study examines the effects of ditch shape modifications on energy dissipation, turbulence intensity, and velocity dispersion [19]. The study helps to optimize ditch design for improved flood management, erosion reduction, and irrigation performance by comparing three case studies with different ditch widths and depths. This work extends the current understanding of ditch-canal interactions by combining CFD simulations with real-world canal conditions [20]. It also validates earlier findings and provides useful information for irrigation planners, canal authorities, and hydraulic engineers working on climate-adaptive water management and infrastructure improvement [21].

III. METHODOLOGY

This study examines the effects of perpendicular ditch shapes on the flow characteristics of a portion of the Abbasia Canal using Computational Fluid Dynamics (CFD) modelling in ANSYS Fluent. Geometric modelling, mesh creation, boundary and hydraulic condition definition, solver and physical model selection, and velocity analysis result post-processing are the main steps of the methodology. A 45-meter segment of the Abbasia Canal, which has a trapezoidal cross-section typical of lined irrigation systems, is the subject of the simulation. The canal's side slopes are made of brick masonry, which adds to the mild wall roughness. The canal's top and bottom widths are 11 and 8 meters, respectively. With a Froude number of 0.1 and an inlet velocity of 0.414 m/s, the flow is presumed to be subcritical and represents typical flow conditions for the research area. Because to upstream loads and natural erosion, there is known to be sediment buildup and fine suspended particles along the canal bed. However, this study does not directly model sediment movement; instead, it focuses on hydrodynamic behaviour (turbulence zones and velocity distribution). Three simulation instances were created for this experiment by altering the depth and width of a perpendicular ditch that was attached to the main canal. The goal is to see how hydraulic performance is affected by these geometrical modifications. All three of the examples that are being examined in this study are shown in Table 1.

Table 1. Simulation cases under consideration for current investigation.

Case	Ditch Width (m)	Ditch Depth (m)	Canal Length (m)	Canal Width (m)	Flow Type	Inlet Velocity (m/s)	Froude Number	Mesh Type
1	3.0	1.5	45	11	Subcritical	0.414	0.1	Triangular
2	3.0	0.75	45	11	Subcritical	0.414	0.1	Triangular
3	6.0	1.5	45	11	Subcritical	0.414	0.1	Triangular

An unstructured triangular mesh was used to discretise the computational domain, allowing for improved conformance to the intricate and curved geometry of the canal-ditch junction. To increase the precision of boundary layer effects and velocity gradients, mesh refinement was used, especially along the ditch edges and adjacent walls.

Grid independence was tested to make sure the results were reliable. The mean flow velocity at control points was used to compare the outcomes of testing three different mesh densities. With roughly 150,000 to 200,000 elements, the chosen mesh provided a mix between computational efficiency and solution correctness. The simulation's boundary conditions were developed using realistic hypotheses that apply to open channel flow. The Abbasia Canal's subcritical conditions were reflected in the specification of the inlet, which had a velocity inlet with a fixed flow speed of 0.414 m/s. A pressure outlet condition was applied at the outlet to promote a smooth flow exit and permit natural water surface variations. In order to simulate the hydraulic resistance imposed by brick masonry linings, the canal walls and ditch boundaries were regarded as no-slip walls. The top barrier was modelled as an open-to-atmosphere pressure outlet in order to authentically depict the open water surface. This allowed for realistic free surface behaviour using the Volume of Fluid (VOF) approach. The Volume of Fluid (VOF) model was used in ANSYS Fluent to simulate the flow's free surface behaviour. This allowed for precise tracking of the air-water interface across the computational domain. To replicate the normal flow regime in the Abbasia Canal, the simulations were carried out in steady-state circumstances with the fluid regarded as incompressible. Gravity effects were introduced in the negative z-direction to account for vertical flow components using a pressure-based solution. The realizable k-ε turbulence model was used to simulate turbulence because it offers accurate predictions for intricate flow structures and energy dissipation, especially in areas where the flow interacts with ditch shapes. Higher numerical accuracy in velocity prediction was ensured by applying the second-order upwind approach to the momentum equations for spatial discretization. The air-water interface was precisely resolved by the geo-reconstruct scheme for the volume fraction equation, and the solution's stable and reliable convergence was achieved using the SIMPLE algorithm for the pressure-velocity coupling. After mass and velocity balance checks verified numerical stability, the simulations were repeated until convergence was reached for all residuals below the 1e-4 criterion. Up to 2,000 iterations were needed in certain situations before a reliable solution was found.

IV. RESULTS AND DISCUSSION

Mean Stream Velocity

The mean stream velocity distribution for Case 1 is shown in Figure 1, which compares the velocity profiles along a vertical segment of the canal before and after the ditch. The y-axis shows the vertical distance (depth) from the canal bed in meters, while the x-axis shows the velocity in meters per second (m/s). The velocity profile before the ditch is represented by the blue line, while the velocity profile after the ditch is represented by the orange line. Furthermore, it is clear that the velocity progressively rises with depth until it reaches peak values for both profiles, which are approximately 0.35 to 0.4 meters. The presence of the ditch, however, intensifies the flow in the top layers of the canal, as seen by the orange line (after the ditch) showing higher velocities in the upper flow zone (above 0.25 m) than the blue line (before the ditch). The velocity for the post-ditch profile drops more precipitously after peaking, indicating that the flow starts to lose energy downstream of the ditch as a result of increased turbulence and possible recirculation zones. In conclusion, the graph shows that the perpendicular ditch in Case 1 causes observable changes in the velocity distribution, particularly in the upper flow zone, where it first speeds up flow but thereafter dissipates more energy. In such systems, where ditches affect streamwise momentum and cause localized flow instability, this pattern is consistent with the expected behaviour.

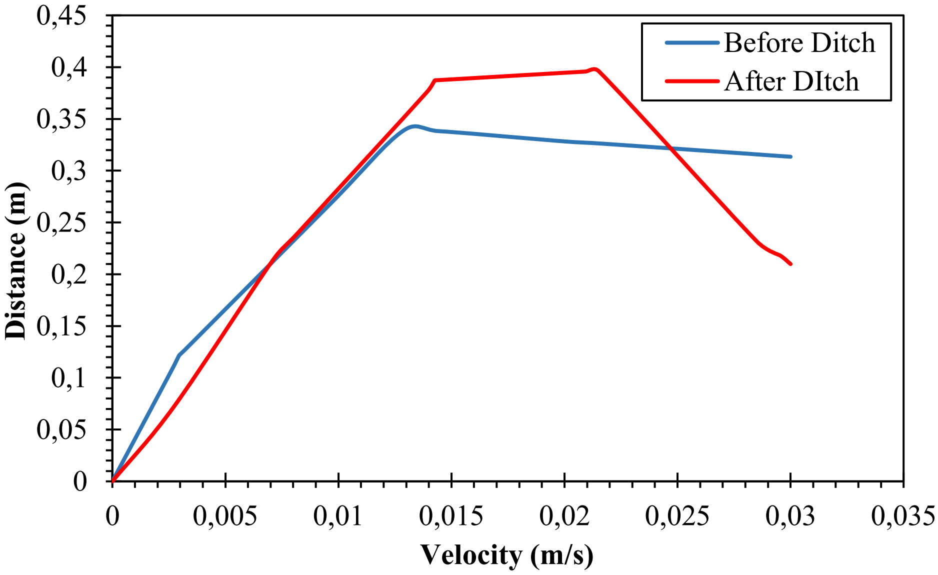


Figure 1. Mean stream velocity for case 1.

The mean stream velocity distribution for Case 2 is shown in Figure 2, which compares the velocity profiles along a shallow vertical portion close to the canal bed before and after the ditch. The vertical distance (m) from the canal floor, up to about 0.035 meters, is shown by the y-axis, while the x-axis displays velocity in meters per second (m/s). The velocity profile before the ditch is represented by the blue line, while the velocity profile after the ditch is represented by the red line. Additionally, it is clear that the velocity profile alters considerably following the ditch. The flow is comparatively steady and concentrated prior to the ditch.

with a steady taper off toward the surface after reaching a peak of 0.31–0.32 m/s. The velocity profile is greater and more dispersed after the ditch, as indicated by the red line, which peaks above 0.4 m/s and continues across the vertical depth. This suggests a slight acceleration and expansion of the flow, as the 0.75-meter-deep ditch in Case 2 raises the mean flow velocity and modifies its vertical distribution. In comparison to deeper ditches, the red curve's smoother and wider profile suggests a more consistent velocity gradient and less flow constriction, which could lead to less intense turbulence. Rather than resulting in sudden energy loss, the modest depth of the ditch seems to promote a more gradual velocity shift. In order to optimize ditch design in irrigation channels, this graph will be helpful in showing how lesser ditch depths can improve flow without causing significant turbulence or energy dissipation.

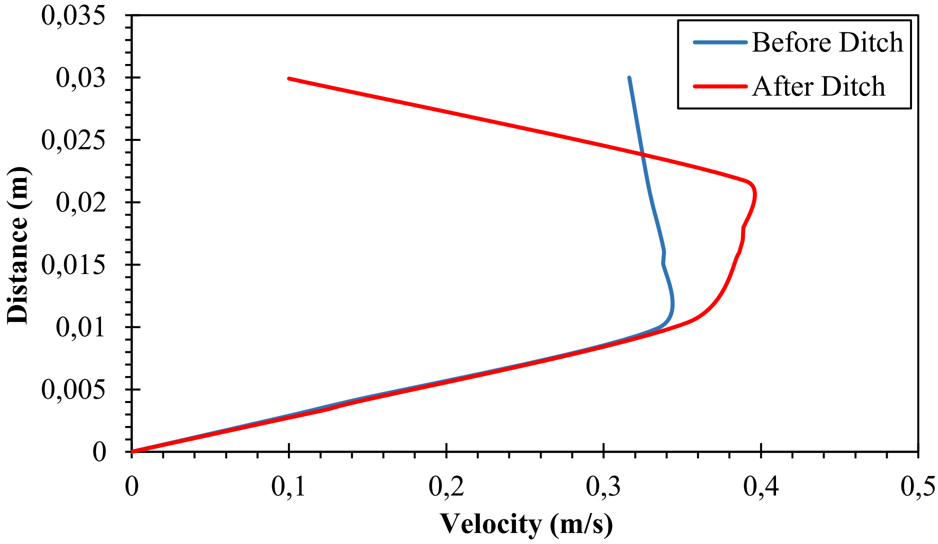


Figure 2. Mean stream velocity for case 2.

The mean stream velocity distribution for Case 3 is shown in Figure 3, which compares the velocity profiles in a vertical portion close to the canal bed before and after the ditch. The distance (m) from the canal floor, up to about 0.035 meters, is shown by the y-axis, while the x-axis displays velocity in meters per second (m/s). The velocity profile before and after the ditch is shown by the blue and red curves, respectively. Compared to the preceding cases, the impact on velocity distribution is much different in Case 3, which has the deepest depth (1.5 m) and the widest ditch (6 m). A substantial, well-developed velocity gradient is suggested by the graph, which shows that the velocity increases continuously with depth before to the ditch, culminating above 0.42 m/s. The red curve, which shows a smaller and lower velocity profile, peaks at about 0.31 m/s. After the ditch, however, the velocity drops off dramatically. This steep drop in velocity following the ditch implies that the deeper and broader ditch absorbs more flow energy, leading to greater dissipation and possibly the formation of larger stagnation or recirculation zones. A broad ditch like this serves as a powerful hydraulic buffer, as evidenced by the post-ditch profile's sharp vertical shift, which also shows flow retardation and energy loss. This arrangement lowers the stream's overall momentum even though it might assist control erosion and slow down high-velocity flows for flood control. The conclusion drawn from this graph is that higher ditch dimensions lead to increased energy dissipation and velocity suppression, which may be advantageous for flow management but, if improperly built, may also result in silt deposition or decreased conveyance capacity downstream.

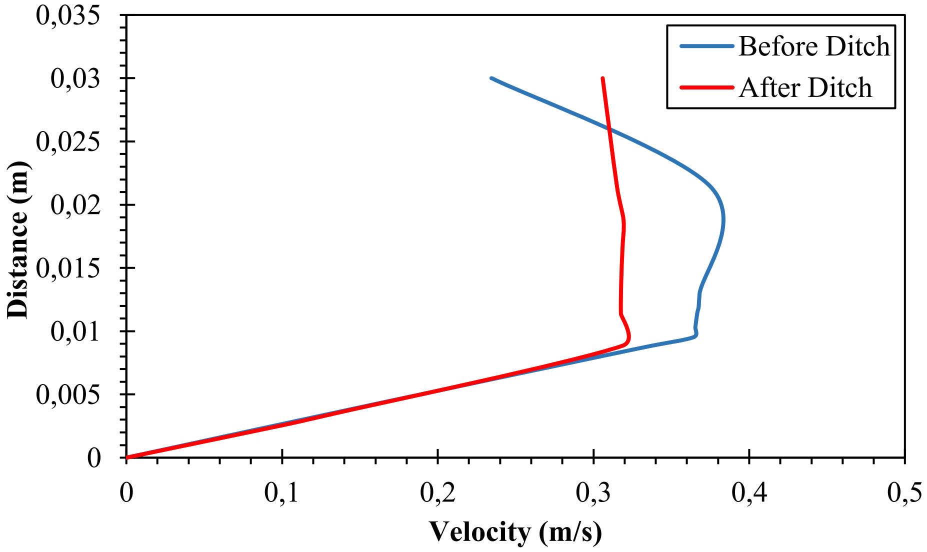
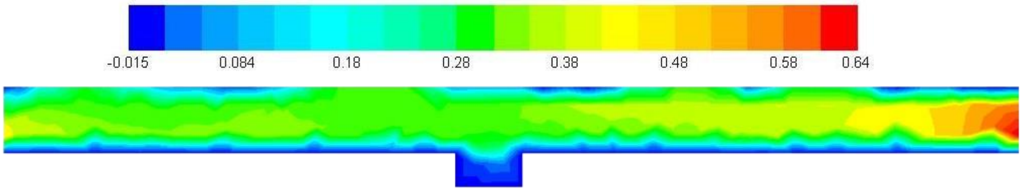


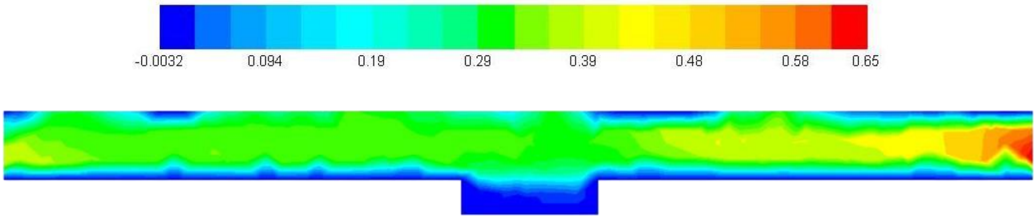
Figure 3. Mean stream velocity for case 3.

Longitudinal Velocity Contours

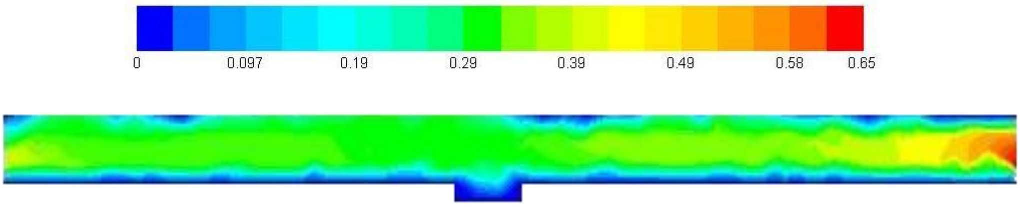
From Figure 4, Important information about how various ditch shapes affect flow behaviour over the Abbasia Canal's length can be seen in the longitudinal velocity contours for Cases 1, 2, and 3. The velocity distribution in Case 1, where the ditch is 3 meters wide and 1.5 meters deep, demonstrates a somewhat uniform upstream flow, with mid-range velocities shown in green hues. Blue and cyan hues indicate energy dissipation as a result of the abrupt increase in depth, which causes a visible slowing as the flow approaches the ditch. The flow re-accelerates downstream of the ditch, and velocity contours exhibit a mixture of yellow and green hues, indicating localized vortex development and significant turbulence. Red zones close to the outflow show that high-velocity pockets are forming as the flow returns..



(a)



(b)



(c)

Figure 4. Longitudinal Velocity Contours; (a) Case 1; (b) Case 2; (c) Case 3.

The velocity curves seem smoother and more streamlined in Case 2, which has a shallower ditch with the same width (3 meters) but a lower depth (0.75 meters). The upstream flow keeps a consistent profile and just slightly slows down when it enters the ditch. The downstream flow rapidly regains velocity without causing much disruption, and the transition through the ditch is constant with little energy loss. The steady green to yellow contour bands that stay tightly aligned and show steady velocity gradients and no turbulence make this clear. Out of the three layouts, Case 2 exhibits the most hydraulically efficient flow behaviour.

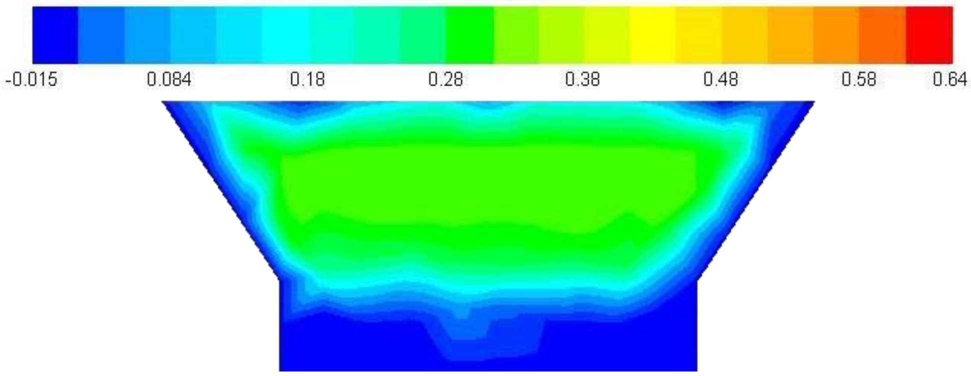
The greatest significant flow interruption, however, is seen in Case 3, which has the deepest and largest ditch (6 meters wide and 1.5 meters deep). An lengthy dark blue zone indicates tremendous energy dissipation and flow expansion as the velocity lowers dramatically when one enters the huge ditch. The contour lines in the downstream region are larger and less compact, and the velocity recovery is slow. Recirculation zones and extended flow separation are suggested by this, which may result in silt deposition and decreased hydraulic efficiency. Although Case 3 effectively manages flow energy and lowers the risk of erosion, its reduced flow momentum impairs discharge capacity. Additionally, it is evident from the longitudinal velocity contours how ditch depth and size affect hydraulic performance. Case 2 is perfect for optimizing water delivery since it provides the smoothest and most effective flow transition with the least amount of turbulence and energy loss. While Case 3 excels at reducing energy but drastically reduces velocity, which might not be appropriate for systems where preserving discharge is a top priority, Case 1 offers a balanced performance with modest dissipation.

Lateral Velocity Contour

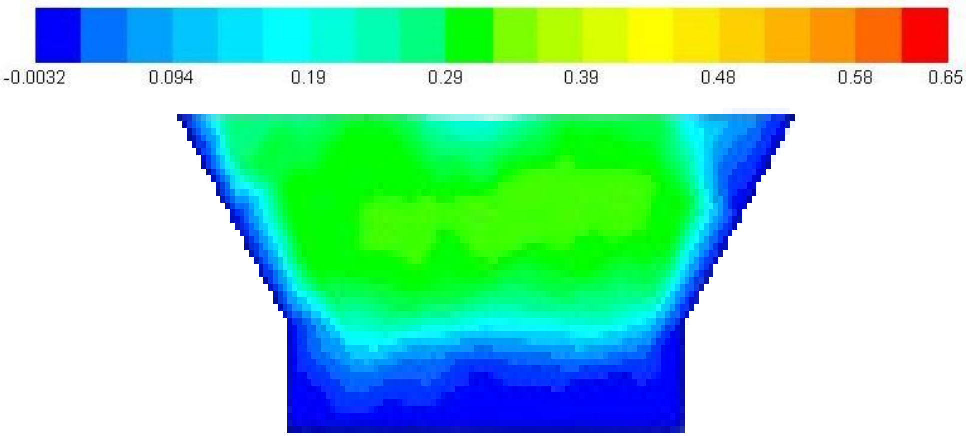
Figure 5's lateral velocity contours for Cases 1, 2, and 3 demonstrate how changes in ditch design affect the Abbasia Canal's cross-sectional flow characteristics. The flow velocity is focused around the center of the cross-section in Case 1, which includes a ditch that is 3 meters wide and 1.5 meters deep. Strong central velocities with a progressive decrease toward the canal edges are indicated by the contours' noticeable green and yellow bands, which range from 0.28 to 0.48 m/s. The deeper ditch implies that the flow is well-focused, but there is still sufficient lateral spread to permit a significant amount of energy dissipation. The profile shows a harmony between controlled turbulence and efficient flow conveyance.

The velocity profile seems more contained and uniformly distributed in Case 2, where the ditch has a lesser depth of 0.75 meters but retains a 3-meter width. Smaller and more concentrated, high-velocity zones are primarily found in the 0.19–0.29 m/s range. The cyan and blue zones at the edges demonstrate how the energy dissipates more rapidly. This promotes hydraulic stability by producing a laminar, smooth flow with little disturbance. Because of the decreased depth, which inhibits excessive vertical flow expansion, Case 2 is advantageous for reducing erosion and guaranteeing consistent water delivery with less turbulence..

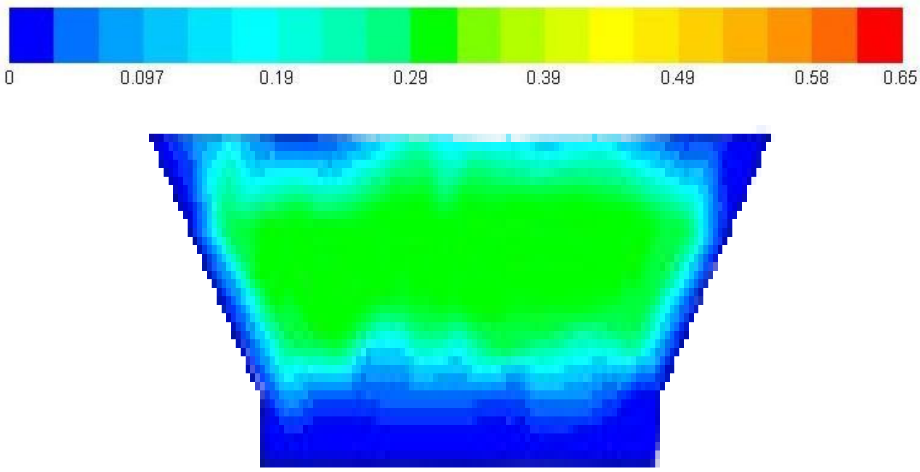
The velocity contours show a wider but less intense flow distribution in Case 3, which has a ditch that is both deeper (1.5 meters) and wider (6 meters). The velocity is less concentrated and more diffused than in the other situations, even though the flow spreads symmetrically throughout the ditch base and side slopes. Lower velocity ranges are shown by the green hues that predominate in the cross-section. This dispersion suggests increased energy dissipation and a notable loss of momentum. Despite being advantageous for reducing flow and halting erosion, this may lower the hydraulic efficiency of the system as a whole and possibly promote silt buildup.



(a)



(b)



(c)

Figure 5. Lateral velocity contours; (a) case 1; (b) case 2; (c) case 3.

All things considered, Case 1 offers a robust, well-channelled flow with a moderate lateral spread that is appropriate for preserving discharge while managing turbulence. For effective and secure water supply, Case 2 offers the most uniform and hydraulically stable profile with the least amount of energy loss. In contrast, Case 3 dissipates energy more effectively by dispersing the flow across a larger region, albeit at the expense of a lower flow velocity. These findings highlight how crucial it is to choose ditch dimensions carefully in order to match the hydraulic performance objectives of the canal.

V. CONCLUSION

It is clear from the thorough examination of longitudinal and lateral velocity contours, velocity profiles, and the effect of ditch geometry on hydraulic performance that ditch dimensions are crucial in determining the flow behaviour of the Abbasia Canal system. Regarding discharge efficiency, energy dissipation, and flow stability, the three scenarios under study show different results. A balanced performance with enough energy dissipation, moderate turbulence, and a well-developed center flow was achieved in Case 1, which had a ditch of moderate width (3 m) and greater depth (1.5 m). This setup maintained a respectable discharge capacity while exhibiting efficient flow management. The most hydraulically efficient behaviour was displayed by Case 2, which had the same width but a lower depth (0.75 m).Both laterally and longitudinally, it showed consistent, smooth velocity distributions with little energy loss or turbulence. This situation is ideal for applications that prioritize consistent discharge and little erosion because the flow easily moved through the ditch. On the other hand, Case 3, which had the deepest and broadest ditch (6 m by 1.5 m), demonstrated a notable loss of energy. It dispersed the flow evenly and successfully decreased peak velocities, but it also increased the likelihood of silt deposition and decreased post-ditch velocity recovery. Instead of optimizing hydraulic efficiency, this scenario can be more appropriate when flow slowing or erosion management are the main objectives. Overall, the analysis finds that while larger, deeper ditches are better for reducing energy but come at the expense of decreased flow efficiency, shallow and moderately sized ditches (like the one in Case 2) provide the optimum balance between discharge capacity and flow stability. These results emphasize how crucial it is to modify ditch shape in irrigation canal systems in accordance with particular hydraulic goals.

REFERENCES

1. Zhao, F., S. Luo, and J. Zhang, Review of the high bed–low ditch system as an alternative to lowland paddy in tropical and subtropical Asia. Frontiers in Sustainable Food Systems, 2024. **8**: p. 1361292.
2. Chow, V., Open channel hydraulics. MacGraw-Hill Book Co. Inc., New York, NY, 1959. **206**.
3. Ahmed, A. and A.R. Ghumman, Experimental investigation of flood energy dissipation by single and hybrid defense system. Water, 2019. **11**(10): p. 1971.
4. Abbas, F.M. and N. Tanaka, Investigation of flow structure with moat acting as a water cushion at the toe of an overflowing levee. Environmental Fluid Mechanics, 2022. **22**(4): p. 865-889.
5. Ramesh, K., F. Mebarek-Oudina, and B. Souayeh, Mathematical modelling of fluid dynamics and nanofluids. 2024: CRC Press Boca Raton, Florida, USA.
6. Waldrip, S., et al., Maximum entropy analysis of hydraulic pipe flow networks. Journal of Hydraulic Engineering, 2016. **142**(9): p. 04016028.
7. Thompson, P.L. and R.T. Kilgore, Hydraulic Design of Energy Dissipators for Culverts and Channels: Hydraulic Engineering Circular Number 14. 2006, National Highway Institute (US).
8. da FRANCA, M.J.R.P., A field study of turbulent flows in shallow gravel-bed rivers. 2005, Verlag nicht ermittelbar.
9. Zaha, T., N. Tanaka, and Y. Kimiwada, Flume experiments on optimal arrangement of hybrid defense system comprising an embankment, moat, and emergent vegetation to mitigate inundating tsunami current. Ocean Engineering, 2019. **173**: p. 45-57.
10. Krafft, D.R., et al., A review of tidal embayment shoaling mechanisms in the context of future wetland placement. 2022.
11. Ahmed, Z., et al., An overview of smart irrigation management for improving water productivity under climate change in drylands. Agronomy, 2023. **13**(8): p. 2113.
12. Nepf, H.M., Flow and transport in regions with aquatic vegetation. Annual review of fluid mechanics, 2012. **44**(1): p. 123-142.
13. Rahman, M.A., N. Tanaka, and N. Anjum, Damming effects of tsunami-borne washed-out trees in reducing local scouring and tsunami energy behind a coastal embankment. Applied Ocean Research, 2022. **126**: p. 103260.
14. Knight, D.W., et al., Practical channel hydraulics: Roughness, conveyance and afflux. 2018: CRC Press.
15. Ferziger, J.H. and M. Perić, Computational methods for fluid dynamics. Vol. 586. 2002: Springer.
16. Sajid, I., et al., Assessing barriers in adaptation of water management innovations under rotational canal water distribution system. Agriculture, 2022. **12**(7): p. 913.
17. Habib, Z. and M. Kuper, Performance assessment of the water regulation and distribution system in the Chishtian sub-division at the main and secondary canal levels. 1998: IWMI.
18. Chiu, C.-L., Velocity distribution in open channel flow. Journal of Hydraulic Engineering, 1989. **115**(5): p. 576-594.
19. Özakin, A.N. and F. Kaya, Effect on the exergy of the PVT system of fins added to an air-cooled channel: A study on temperature and air velocity with ANSYS Fluent. Solar energy, 2019. **184**: p. 561-569.

20. Gandhi, B., H. Verma, and B. Abraham. Investigation of flow profile in open channels using CFD. in 8th Intl Conference on Hydraulic Efficiency Measurement. IGHM. 2010.
21. Sahoo, A., S. Samantaray, and R.B. Singh, Analysis of Velocity Profiles in Rectangular Straight Open Channel Flow. Pertanika Journal of Science & Technology, 2020. **28**(1).