ZANCO Journal of Pure and Applied Sciences

TYPE Original article
DOI http://dx.doi.org/10.21271/ZJPAS.37.4.14

OPEN ACCESS

*Corresponding author

Azheen Karim Fatah

RECEIVED :05 /02 /2024
ACCEPTED :14/04/ 2025
PUBLISHED :31/ 08/ 2025

KEYWORDS:
modeling, simulation, piers, Flow-3D, velocity profile.

Influence of Pier Geometry on Flow Characteristics using CFD

Azheen Karim Fatah* ,Abdulla Abdulwahid Abo ,Pshtiwan Othman Zaid and Arkan Hamza

Department of Water Resources Engineering, College of Engineering, Salahaddin University-Erbil, Erbil, Kurdistan Region, Iraq.

ABSTRACT

An important part of hydraulic engineering that is vital for the security and durability of bridge structures understands the velocity of water flow around bridge piers, and this is what this research aims to do. The flow characteristics around three different pier shapes—triangular, rectangular, and circular—were the primary focus of the investigation. The study tested the computational model against experimental data from a reputable source to make sure it was accurate. A discrepancy of less than 10% was found between the model's predictions and the actual experimental data, indicating a good degree of agreement. The results showed that the maximum flow rates just before the center piers varied with pier shape. The highest velocity recorded by the rectangular pier was 0.592 m/s, while the round and triangular piers reached 0.553 and 0.550 m/s, respectively. As one moved farther downstream, the velocity of all designs dropped significantly. At 0.261 m/s, the rectangular pier was the fastest, followed by the triangle at 0.256 m/s, and finally the round at 0.249 m/s. According to the findings, the smoother form of the circular pier is more successful at decreasing flow disturbance than that of the rectangular pier, which has sharper edges and a larger frontal surface and so disrupts the flow more. Findings from this research are crucial for hydraulic engineers because they show how to choose the right pier designs to increase the stability and longevity of bridges without altering the natural flow conditions.

1.Introduction

There has been a lot of study on the topic of scour and flow behavior around bridge piers, but much of it has ignored the comparative effects of common geometries like triangles and rectangles in favor of cylindrical or conventional pier designs. In addition, there has been a dearth of thorough numerical simulations using verified computational fluid dynamics (CFD) models to assess and contrast the hydrodynamic behavior around different pier designs, despite the extensive use of empirical and experimental approaches. Engineers are unable to maximize hydraulic performance and structural stability in pier design due to this mismatch. So, to address this knowledge gap, this work used verified CFD modeling to examine how different pier shapes (circular, rectangular, and triangle) affect flow velocity profiles. The findings will provide useful information for sustainable bridge design. According to research done by (JASIM et al., 2020), the cylindrical pier works better in areas with less interference with the water flow. As an alternative to cylindrical piers, hexagonal and square piers are provided on the second level. These may be used in cases when additional protection for the foundations is necessary due to an immediate architectural problem. When a structure is built on top of an easily-eroded bed, it increases the local movement of sediment, which causes the region around the structure to be eroded. Local scouring near bridge piers is particularly dangerous for hydraulic bridges, even if scouring happens naturally (Najafzadeh and Barani, 2011). When it comes to coastal, offshore, and bridge engineering, local scour is a major cause of structural failures. The research indicates that pier scour has recently been the cause of many bridge collapses (Wardhana and Hadipriono, 2003). The effective pier width, which is equal to the diameter of the smallest circumscribing circle of the pier, determines the equilibrium dimensionless scour parameters, which include scour depth, length, breadth, area, and volume (Das and Mazumdar, 2014). When the approaching flow depth remains constant and the pier diameter rises, the scour characteristics become more pronounced. The following scour parameters—depth, length, breadth, area, and volume—are estimated using empirical equations supplied by the findings. The scour depth around a bridge pier's curvature is increased due to the flat exposed frontal part of the pier creating an extremely strong horseshoe vortex (Baranwal et al., 2023). The smooth curve of the pier divides the flow, which in turn reduces the strength of horseshoe vortex generation, and so the scour depth around the sharp nose shape is reduced. Based on their findings, the researchers concluded that incoming flow velocity and exposed frontal area of the bridge pier are closely correlated with the depth of scour around a pier. Finding the best form for bridge piers is expected to be made easier by the study's results. Because of the increased shear stress caused by this buildup, the flow's ability to move silt is enhanced. According to (Ali and Karim, 2002), the principal scouring agent is the downhill flow that hits the bed. There is more evidence from experimental investigations of scour around a vertical circular cylinder than from numerical research. The available computing power has been a major constraint for the scour research community, which is the main reason for this. One benefit of numerical approaches is that they may be used at the prototype size without being affected by scaling effects. However, in order to account for sediments, numerical models need to use empirical equations. It was published by (Zhao et al., 2010). (Zaid and Mamand, 2024) conducted research Both computational and experimental analyses have shown that, for two meters upstream of the steep crested weir, the water surface will remain relatively flat, and that, due to the weir's curved shape, longitudinal water levels will fall little. (Zaid and Abo, 2022) run 45 simulations with a constant chute slope of 26 degrees, three different step angles, three different step numbers (20, 30, and 40), and five different skimming flows. In general, when both the flow rate and the steep slope are big, increasing the steep slope has a greater impact on the inception point length than reducing the step height. However, when both the flow rate and the steep slope are large, increasing α has no effect on the inception point length. The inception point in the flat stepped spillway model is closer to the spillway crest when compared to the sloping stepped spillway model with the same flow. For a given flow, the inception point length of a flat and a slopped stepped spillway is same; however, the difference becomes apparent for bigger discharges and becomes more pronounced as the flow increases. It may be inferred from (Abo, 2022) that wide crested weirs are less effective in discharging flow than sharp crested weirs because to their lower discharge coefficient. Broad crested weirs are less likely to experience cavitation due to their lower cavity index compared to sharp crested weirs. For the free surface flow over a broad-crested weir, (Hargreaves et al., 2007) compared a set of experimental data with a series of computational fluid dynamics (CFD) simulations. Reproducing the experimental free surface profiles, pressure and velocity profiles, and discharges across the weir for a variety of discharge rates was achieved by setting the upstream and downstream water depths in the CFD model. The findings are shown to be quite sensitive to the turbulence model choice.

2.Methodology

For fluid dynamics, the Flow3D model is an unrivaled strength. A one-, two-, or three-dimensional flow field may be studied using this model. Its range of uses in the field of fluid dynamics is extensive. The Navier-Stokes equations form the basis of this model; five different turbulence models are used to solve the features of turbulent flow.

The dynamics of fluid compounds, including gases and liquids, are governed by the Navier-Stokes equations. The complex dynamics of fluid flow in three dimensions and over time may be well described using tensorial formulations. The tensor form of the Navier-Stokes equations looks like this:

Continuity Equation (Conservation of Mass):

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho u^{i}\right)}{\partial x^{i}}=0 \qquad (1)$$

Where: ρ is the fluid density and u^i is the velocity component in the i - direction.

Momentum equation

$$\frac{\partial\left(\rho u^{i}\right)}{\partial t}+\frac{\partial\left(\rho u^{i} u^{j}\right)}{\partial x^{j}}=-\frac{\partial p}{\partial x^{i}}+\frac{\partial T^{ij}}{\partial x^{j}}+f^{i} \qquad (2)$$

Where:

- p is the pressure,
- T is the viscous stress tensor
- f represents body forces (e.g., gravity),

Energy Equation

$$\frac{\partial \rho e}{\partial t}+\frac{\partial\left(\rho e u^{i}\right)}{\partial x^{i}}=\frac{\partial}{\partial x^{i}}\left(k \frac{\partial T}{\partial x^{i}}\right)+T^{ij} \frac{\partial u^{i}}{\partial x^{j}}+f^{i} u^{i} \qquad (3)$$

Where:

- e: specific internal energy,
- k: thermal conductivity,
- T : temperature

3.Code validation

The experimental findings from (Abo, 2013) were used for model validation. Figures 1 and 2 show the trapezoidal cross-sectional channel, which has a bed width of 0.6096m, a bed slope of 0.0035, and side slopes of 1:2.25. The channel also has three piers, each with rounded ends, measuring 0.3048m in length and 0.03048m in thickness, and they are located 9.144 m from the upstream border. In the absence of piers, with $Q = 0.1133$ m^3.s^{-1}, $n = 0.009$ (for plywood and glass materials, $Fr = 1.4$, by using the Manning equation, the normal depth is computed to be 0.1068 m.

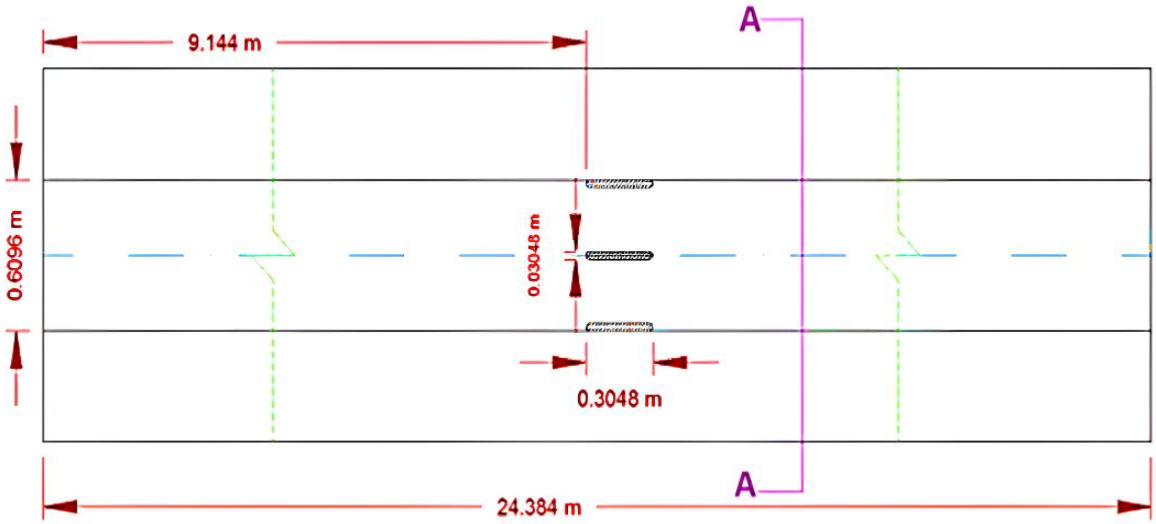

Figure 1: Plan

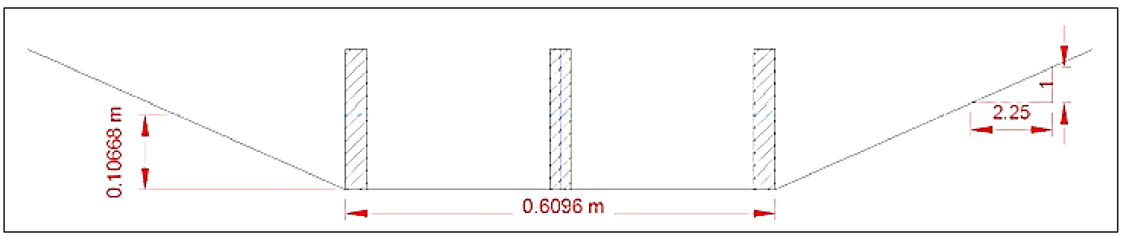

Figure 2: cross section

The fact that the highest disparity at any given point does not exceed 10% explains the good agreement shown between both results of experimental and numerical, Figure 3.

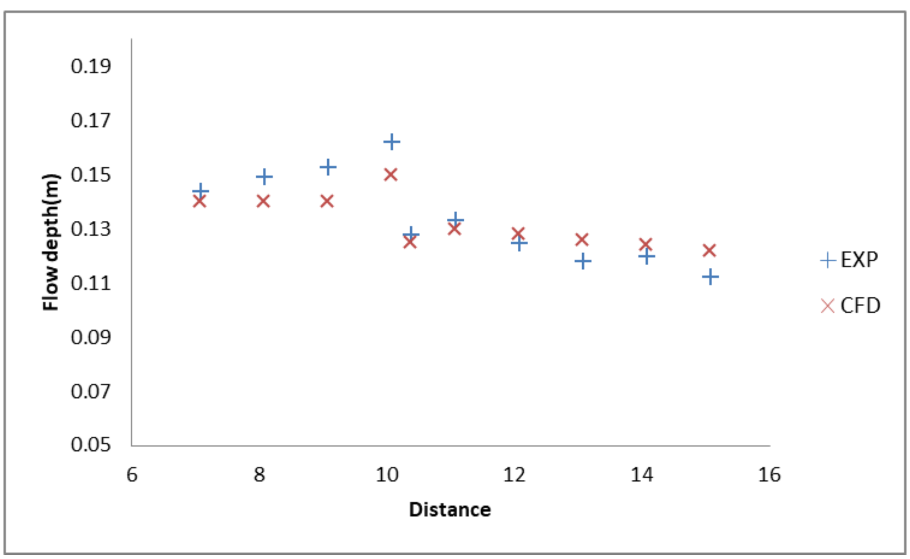

Figure 3: CFD and EXP results

Error Metrics for Model Validation

To validate the CFD model against the experimental data (from Abo, 2013), the following standard error metrics were used:

Table 1. Comparison of Experimental and CFD Velocity Values

Observation Point	Experimental Velocity (m/s)	CFD Velocity (m/s)
1	0.56	0.52
2	0.59	0.57
3	0.61	0.60
4	0.58	0.54
5	0.57	0.55

1. Mean Absolute Error (MAE):

$$MAE = \frac{1}{5}\sum_{i=1}^{5}(V_{CFD,i} - V_{CFD,i})$$

(4)

$$MAE = \frac{1}{5}(0.04 + 0.02 + 0.01 + 0.04 + 0.02) = 0.026\ \mathrm{m/s}$$

2. Root Mean Square Error (RMSE):

$$RMSE = \sqrt{\frac{1}{5}(V_{CFD,i} - V_{CFD,i})^{2}}$$

(5)

RMSE=

$$\sqrt{\frac{1}{5}(0.0016 + 0.0004 + 0.0001 + 0.0016 + 0.0004} = \sqrt{0.00082} \approx 0.0286\ m/s$$

3. Mean Absolute Percentage Error (MAPE):

$$MAPE = \frac{\%100}{5}\sum_{i=1}^{5}\left[\frac{V_{CFD,i} - V_{EXP,i}}{V_{EXP,i}}\right]$$

(6)

MAPE=4.5<10 good agreement with experimental

The flow around piers has been modeled in this study using the RNG turbulence model. Utilizing the same model used during validation, three distinct shapes—a circle, a triangle, and a square—were modelled. We went with these designs since they are so common when building bridge foundations.

Boundary condition:

Upstream of the bridge pier is where the inflow, which is a volumetric flow rate, is located. At the further end of the system, downstream from the bridge piers, there is a pressure outlet that serves as the boundary condition. See Figure 4 for an illustration of how the top boundary is defined as a symmetry requirement and how the left, right, and bottom borders are created as walls.

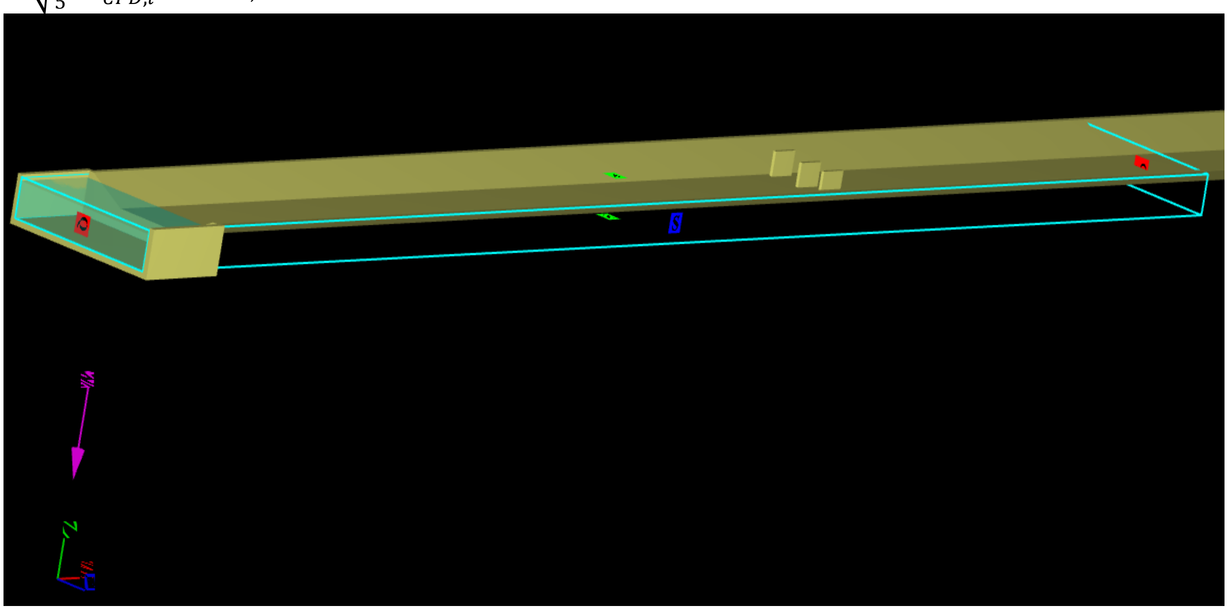

Figure 4: boundary conditions

Mesh Sensitivity and Refinement:

In this study, a uniform mesh with approximately 5 million cells was applied throughout the entire computational domain. The selected mesh resolution was finer than the thinnest geometric features of the model, particularly the narrowest sections of the bridge piers; it guaranteed that accuracy is high for all points inside the model not just near the piers. This ensured that the flow characteristics around the piers were captured with sufficient accuracy even it requires more time to run the scenario. Although local mesh refinement was not used, the fine global mesh was deemed adequate based on a mesh convergence assessment and validation against experimental data. The comparison between CFD and experimental results yielded a mean absolute percentage error (MAPE) of less than 5%, confirming that the selected mesh resolution was both efficient and accurate for the objectives of this study.

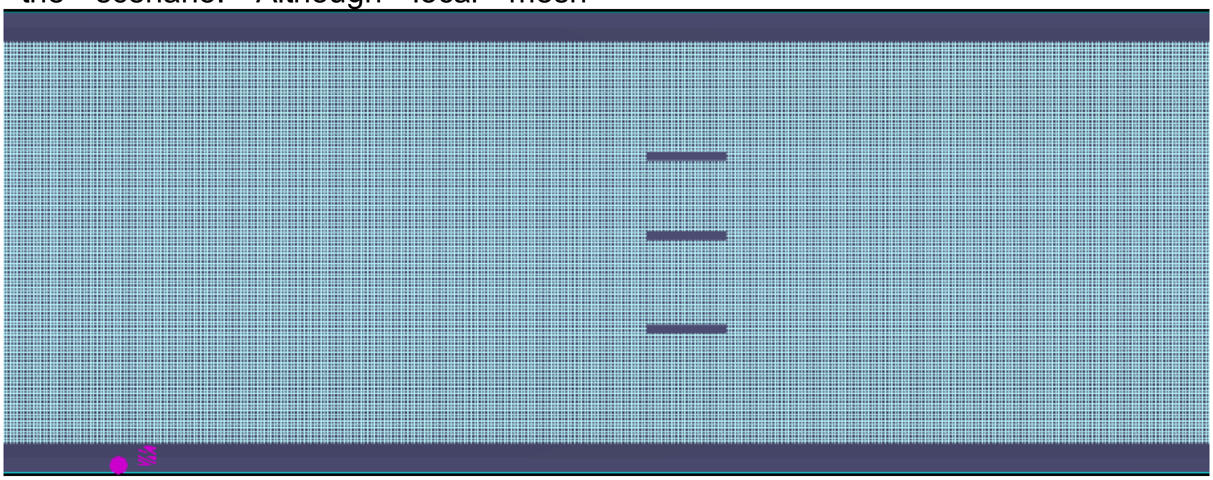
Figure 5: Mesh size

4.Results and Discussion

A crucial component of hydraulic engineering that guarantees the safety and longevity of bridge constructions is the velocity of water flow around the piers. This research aimed to comprehend this flow. Examining the flow characteristics around three distinct pier shapes—circular, rectangular, and triangular—was the primary objective. The impact of the piers' geometry on the river's velocities was the subject of an inquiry. Research showed that different shapes resulted in different flow velocities upstream of the intermediate piers. The rectangular pier reached a maximum speed of 0.592 m/s, situated 5 cm upstream. The highest speeds for the triangular pier were 0.553 m/s, whereas for the circular pier they were 0.550 m/s. These results indicate that, in comparison to the round and triangular piers, the rectangular pier, with its sharper corners and larger front surface area influences the flow more significantly. The maximum velocities for all forms were significantly reduced after passing the middle piers. In terms of maximum velocity, the rectangular pier came in at 0.261 m/s, the triangular pier at 0.256 m/s, and the circular pier at 0.249 m/s. The water loses some of its flow energy as it goes around the piers, which is why the speeds downstream are lower. Circular piers are efficient in decreasing flow disturbance because their streamline design results in the lowest downstream velocity.

To summarize, the pier's rectangular design significantly affected the local flow conditions, as it continuously displayed greater maximum velocities in both the upstream and downstream directions. In terms of minimizing flow disruption, the circular pier proved to be more efficient than the triangular one, which had middling values. This study's results will greatly impact hydraulic engineering and pier design for bridges. They are a great resource for engineers looking to keep natural flow conditions intact while constructing stable and long-lasting bridges by determining the best pier shapes.

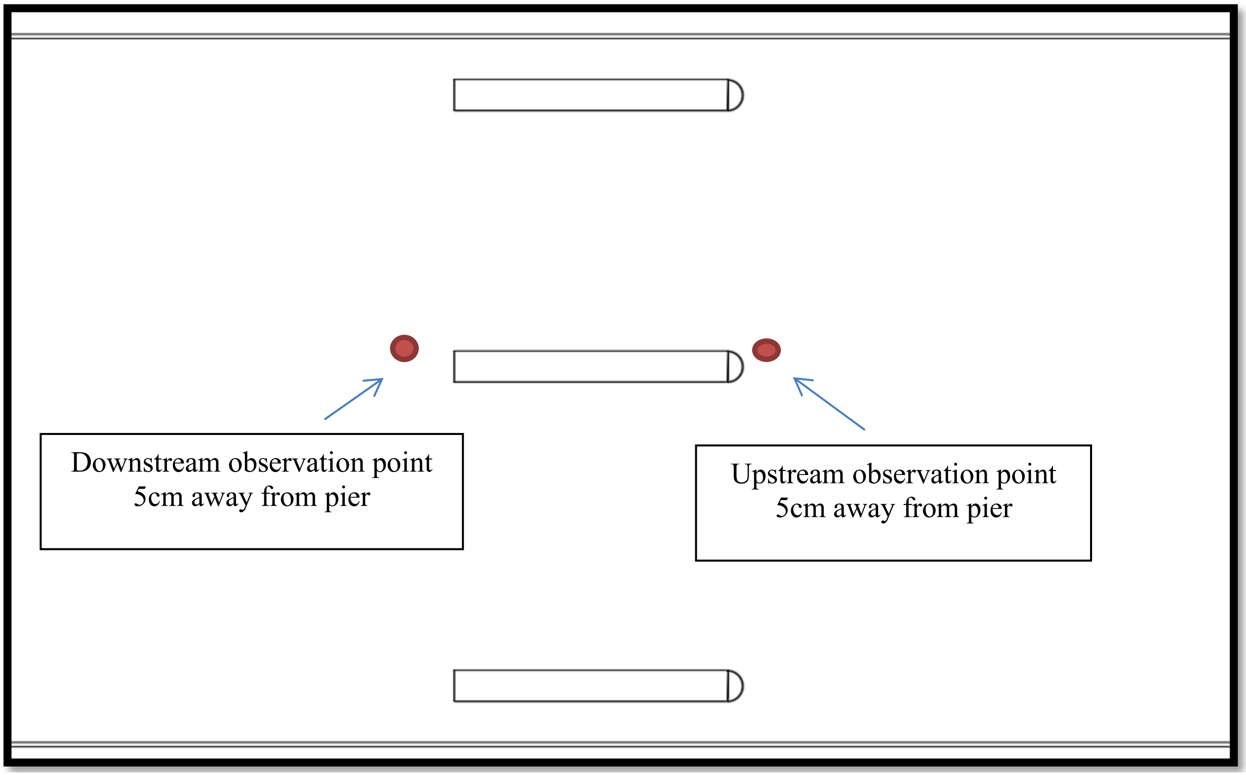

Figure 6: result locations

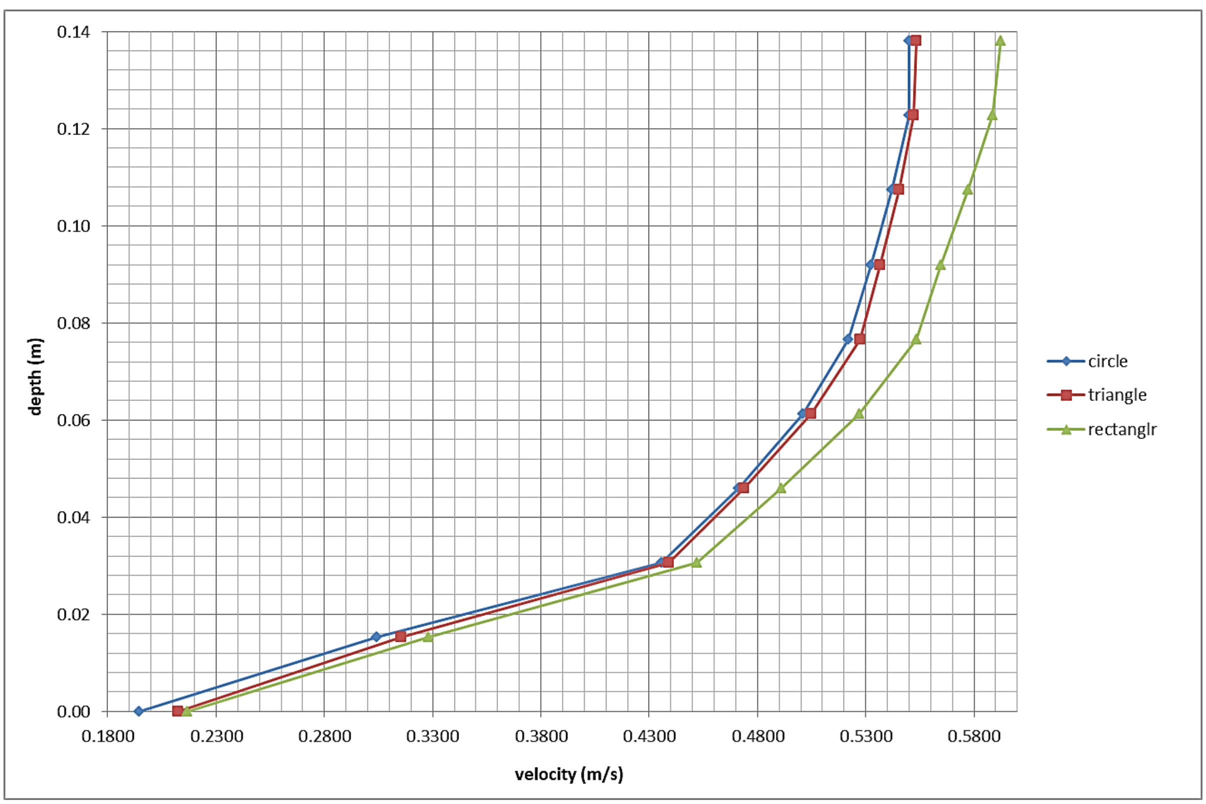

Figure 7: CFD, upstream velocity profile

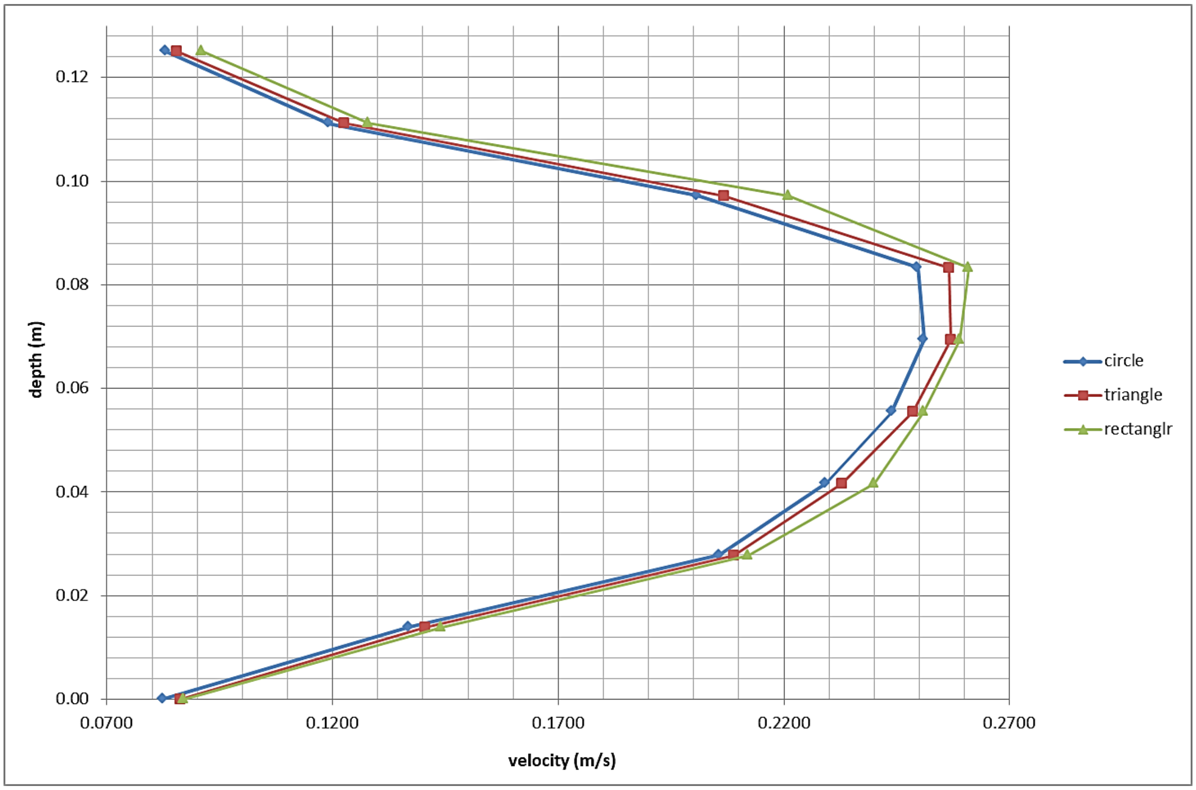

Figure 8: CFD, downstream velocity profile

5.Conclusion

This research highlights the critical influence of pier geometry on the flow characteristics around bridge structures. Through validated CFD simulations using a high-resolution mesh of 5 million cells, the study examined how three commonly used pier shapes—rectangular, triangular, and circular—affect upstream and downstream velocity distributions in an open channel. The results clearly indicate that rectangular piers cause the greatest disturbance to flow, with the highest recorded velocity upstream (0.592 m/s) and downstream (0.261 m/s). This is attributed to their sharp corners and broad frontal surface, which increase resistance and intensify local turbulence. In contrast, the circular piers demonstrated the most hydrodynamically efficient behavior, producing the lowest upstream (0.550 m/s) and downstream (0.249 m/s) velocities. The triangular piers showed moderate performance, slightly more disruptive than circular piers but less so than rectangular ones. These differences are not only statistically significant but also practically important. The smoother, more streamlined shape of circular piers reduces flow separation and vortex formation, which are known contributors to local scour and structural degradation over time. Therefore, adopting such geometries can improve the hydraulic performance, reduce sediment transport impact, and enhance the structural longevity of bridge foundations.

Overall, this study provides a strong recommendation for the use of circular piers in locations where minimizing flow disruption and scour is a priority. The methodology and validated model used in this research can serve as a useful framework for future design evaluations and optimization of bridge pier shapes in varying hydraulic environments.

Funding: This research did not get any funding from external sources.

Conflicts of Interest: The authors assert that they have no conflict of interest.

Acknowledgement: The authors gratefully acknowledge the support provided by Salahaddin University-Erbil in facilitating this research.

6.References

ABO, A. A. 2013. A three-dimensional flow model for different cross-section high-velocity channels. *Doctoral dissertation, University of Plymouth*.

ABO, A. A. 2022. Performance Against Cavity Index and Discharge Coefficient between Broad and Sharp Crested Weirs. *Polytechnic Journal*, 12(2): 45–56.

ALI, K. H. & KARIM, O. 2002. Simulation of flow around piers. *Journal of hydraulic research*, 40(1): 25–34.

BARANWAL, A., DAS, B. S. & SETIA, B. 2023. A comparative study of scour around various shaped bridge pier. *Engineering Research Express*, 5(3): 035001.

DAS, S., DAS, R. & MAZUMDAR, A. 2014. Variations of clear water scour geometry at piers of different effective widths. *Turkish Journal of Engineering and Environmental Sciences*, 38(4): 567–578.

HARGREAVES, D. M., MORVAN, H. P. & WRIGHT, N. G. 2007. Validation of the Volume of Fluid Method for Free Surface Calculation: The Broad-Crested Weir. *Engineering Applications of Computational Fluid Mechanics*, 1(2): 136–146.

JASIM, S. D., AL-DABBAGH, HASEN, A. T. & SHEKHO, A. A. 2020. Evaluation Of Flow Behavior Around Bridge Piers. *Solid State Technology*, 63(6): 3214–3225.

NAJAFZADEH, M. & BARANI, G. A. 2011. Comparison of group method of data handling based genetic programming and back propagation systems to predict scour depth around bridge piers. *Scientia Iranica*, 18(5): 1089–1095.

WARDHANA, K. & HADIPRIONO, F. C. 2003. Analysis of recent bridge failures in the United States. *Journal of performance of constructed facilities*, 17(3): 144–150.

ZAID, P. O. & ABO, A. A. 2022. Determination the Location of an Air Inception Point for Different Configurations of Stepped Spillways using CFD. *Zanco Journal of Pure and Applied Sciences*, 34(3): 45–57.

ZAID, P. O. & MAMAND, B. S. 2024. The Validation of Rectangular Sharp Crested Weir Flow using ANSYS–FLUENT. *Zanco Journal of Pure and Applied Sciences*, 36(2): 112–120.

ZHAO, M., CHENG, L. & ZANG, Z. 2010. Experimental and numerical investigation of local scour around a submerged vertical circular cylinder in steady currents. *Coastal Engineering*, 57(8): 709–721.