CONSTRUCTION
E-ISSN: 2785-8731
VOLUME 5, ISSUE 2, 2025, 247 - 254
DOI: https://doi.org/10.15282/construction.v5i2.12211

RESEARCH ARTICLE

Investigation of Water Flow Behaviour Around Circular Bridge Pier Using Computational Fluid Dynamics

S. N. H. Mohd Yusof[1]*, T. R. Md Kassim[1], N. Z. Nik Azizan[2], N. Md Husain[1], N. K. Basri[1]

[1]Department of Civil Engineering, Kulliyyah of Engineering, International Islamic University Malaysia, P.O. Box 10, 50728, Selangor, Malaysia

[2]Faculty of Civil Engineering & Technology, Universiti Malaysia Perlis, Kompleks Pusat Pengajian Jejawi 3, 02600, Arau, Perlis, Malaysia

ABSTRACT - Structure resistance such as bridge pier to flooding is a critical parameter that should be taken into consideration in analysis and design to prevent any structure failures mainly under natural disasters. Moreover, flood phenomenon has the ability to cause a turbulence flow that leads to increments of water velocity that may impact the bridge piers. Thus, the main objective is to investigate and compare the water flow behaviour around a circular bridge pier with and without fender installations, using Computational Fluid Dynamics (CFD). In order to study performance of circular pier under the influence of water flow, single phase Reynolds-Averaged Navier-Stokes (RANS), Finite Volume Method (FVM), k – epsilon and k – Omega turbulence models have been adopted in the simulation to reflect the boundary turbulence conditions. The outcome of the study indicates that installation of barriers managed to reduce adverse pressure gradient formation, leading to lesser strength of horseshoe vortex of the water flow behaviour surrounding the circular pier. In conclusion, CFD simulation tool offers effectively analyses and understands water flow behaviour around circular bridge piers, both with and without barriers installed in current study.

ARTICLE HISTORY

Received	:	24 March 2025
Revised	:	15 May 2025
Accepted	:	30 June 2025
Published	:	29 December 2025

KEYWORDS

Circular bridge pier
CFD
Water flow behaviour
Barrier

1. INTRODUCTION

Safety of bridges and structures safety during extreme weather such as floods and storms should be addressed critically since these structures may lose their stability due to natural disaster impacts. One of the main causes of bridge failures is due to the floods phenomenon when the water suddenly turns turbulence as the water impact increases on the surrounding of bridge piers. Mitoulis et al. [1] stated that the biggest threat to infrastructure and bridges is flooding. This infers that the damage to the bridge's structure will suffer greatly when subjected to flood impact. Gautam and Dong [2] stated that floods can be disastrous to bridge structures. The resulting damage of the bridges may disrupt the transportation system and lead to economic loss as it takes time to recover. Flood induced bridge failure can be attributed to scouring, deterioration, sedimentation, water pressure, hydrodynamic forces, debris impact, and log impact [3-4]. Liang et al. [5] explained that due to the presence of the pier, the flow is blocked, and the velocity is at complete rest (equal to 0) which results in an increment of water pressure at the pier surface. Gradually, the velocity and the pressure at the pier surface will decline and create a pressure gradient along the stagnation line. Then, the flow is forced to drive downward causing erosion and induced a scour hole at the pier base. A helical vortex called horseshoe vortex is formed within a scour hole due to the interaction of downward flow and incoming flow as shown in Figure 1. Singh et al. [6] stated that the stagnation line will accelerate the flow by the side of the pier. The separation of flow occurring around the pier will create the wake vortex which flows as tornados that lift the sediments and put it in suspension away from the base pier region [7]. As the vortex gets distant from the pier, the sediments start to be deposited downstream of the pier due to the drastic depletion of the vortices' intensity. Even though both vortices cause scours, their critical differences to each other with horseshoe vortex has a more severe effect. The existence of bridge piers not only significantly influence the scour hole formation but also the water velocity and fluid pressure distribution during floods. As claimed by Bombar [8], the presence of piers causes steeper velocity profiles and reduces stream-wise velocities, with larger piers causing greater reductions. The change in flow pattern around the pier is influenced by the pier Reynolds number. Huang et al. [9] mentioned that with the presence of guide panels will reduce the scour depth by 90%. On the other hand, Vaghefi et al. [10] and Ning et al. [11] used spur dikes and the water flow will be weakening.

Misuriya et al. [12] investigated the flow field around a pier with a circular section on a gravel bed. To assess the impact of flow depth relative to the pier diameter, the study analysed vorticity, Reynolds shear stress, and turbulent kinetic energy. Grioni et al. [13] employed computational fluid dynamics to examine the flow around a cylinder in the vicinity of a solid surface. Ramos et al. [14] mentioned that studies on flow on a circular cylinder shape is relevant in many engineering fields. The flow disturbance around a circular cylinder tends to induce scours around the structures. For many years, due to the complexity of the flow structures conducted in experiments, the numerical modelling of the flow around piers has been gaining attention. The authors suggested that further quantitative numerical models will be beneficial compared with the efforts in the experiments.

***CORRESPONDING AUTHOR | Tahara RMK** | ✉ tahara@iium.edu.my

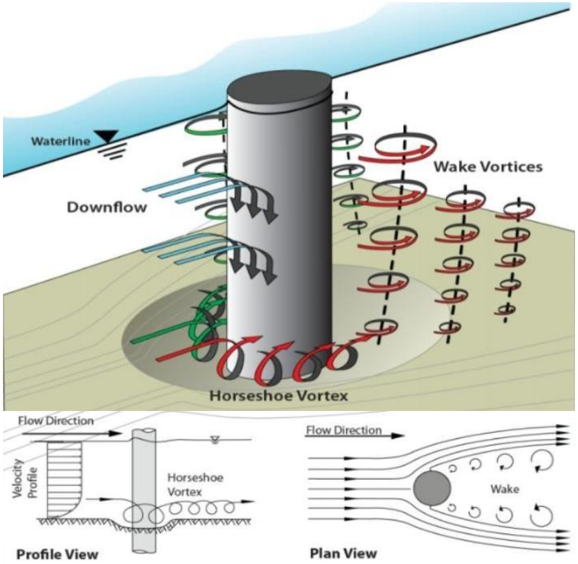

Figure 1. Flow pattern on circular bridge pier [7]

Ghadeeri and Abbasi [15] had performed a study numerically by using FLOW–3D model to investigate local scour around airfoil-shaped pier with collar. FLOW-3D models utilize the volume of fluid (VOF) method to analyse three-dimensional flow and perform free surface modelling by solving Reynolds averaged Navier- Stokes equations. Vaghefi et al. [10] stated that installation of spur dike reduced the scouring by 50% compared to a case when no spur dike is installed. Ning et al. [11] conduct the experiment for a grouped spur dikes with different spacing on a straight path and identified that the more spacing between the spur dikes will weaken the shielding effect. Wu et al. [16] simulated the laminar flow over a stationary circular cylinder with a flapping plate, using a Reynolds number of 100. The wake flow at the rear of the cylinder and plate was strongly influenced by the amplitude and frequency of the plate's motion. Additionally, the presence of the flapping plate altered the flow structure. Sowoud et al. [17] conducted a numerical simulation to evaluate the use of the standard (k-ε) model and investigate the impact of Reynolds number on the flow around a circular cylinder. Jasim et al. [18] mentioned that mentioned that the flow of water is significantly influenced by the geometrical shape of the pier, with cylindrical piers showing the best performance by causing minimal disturbance to the flow. Recent studies have focused on the turbulence characteristics that contribute to scour around circular, tandem, staggered, and oblong pier arrangements [19-20]. Thus, the main objective is to investigate and compare the water flow behavior around a circular bridge pier with and without fender installations opting Reynolds-Averaged Navier-Stokes (RANS), Finite Volume Method (FVM), k – epsilon and k – Omega turbulence model. The flow behaviour around a 2-dimensional (2D) cylindrical pier in a flat fixed bed, representative is further studies to achieve aims of current studies.

2. METHODOLOGY

Current study focused the flow behaviours around circular bridge pier that is modelled by using ANSYS Fluent software by opting for approaches such as Finite Volume Method (FVM). There are two cases of model being considered where the first case is a circular pier subjected to water flow. Meanwhile the second case is barriers placed at the upstream of the circular pier. Further details of the simulation modelling and validation were based on studies by Prasanna and Kumar [21]. Current study adopted the Reynolds-Averaged Navier-Stokes (RANS) equations as a flow solver (water flow). This approach involves solving two differential equations where one represents the generation of transport of turbulence while the other represents the transport of dissipation of turbulence. Current study uses a single-phase flow model that employs standard k – ω models to reflect the turbulence boundary conditions. The adopted models are Turbulence Kinetic Energy (k) and the Dissipation Rate (omega, ω) since sediment transport and scour around the pier are not included in the numerical simulation.

2.1 2D structure modelling using Ansys Fluent (Case 1)

Figure 2 shows structure modelling executed by referring to the geometry details obtained from Prasanna and Kumar [21] studies. A 2D geometry of circular pier was modelled with 0.025 m diameter (scaled down) in a flat fixed bed. Next, mesh was generated by using edge sizing and refinement methods were considered in the analysis. The edge sizing number of divisions used is about 50 around the circular pier. Meanwhile, the refinement method is used in the mesh to accurately simulate the surrounding of the pier using both quadrilateral and triangle meshing types. It is well known that small mesh sizing will produce more accurate results during calculation. Thus, a coarser mesh was adopted away from the pier boundaries to cover the whole domain. This is due to compute and accurately capture the surrounding of the pier. Figure 3 shows the mesh generation implemented in current study. Next, all sections such as inlet, outlet, channel walls, and piers are created as named selection. Figure 4 shows four sections have been created based on Prasanna and Kumar [21] studies. Before running the analyses, the solver type is set as pressure-based while the time is set as transient. Next, under the boundary conditions, all parameters for inlet, outlet, channel walls, and pier are set up according to the information as tabulated in Table 1. Roughness constant is taken into consideration since it is a parameter that accounts for the uniformity of the particles on the surface that varies between 0.5 to 1, the value of roughness constants is set as 0.5 to indicate a uniform surface on the channel walls. A value of 0.041 m/s is set up as the mean velocity of the water (water flow) along the channel from upstream to the downstream. Lastly, for the fluid material, water-liquid (H2O-l) is chosen and the model is completely ready to be analysed to achieve the objective.

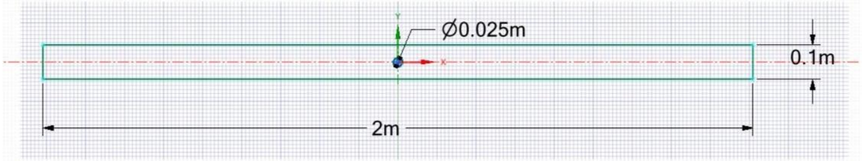

Figure 2. Geometry of the circular pier for Case 1

Figure 3. Meshing generated for Case 1

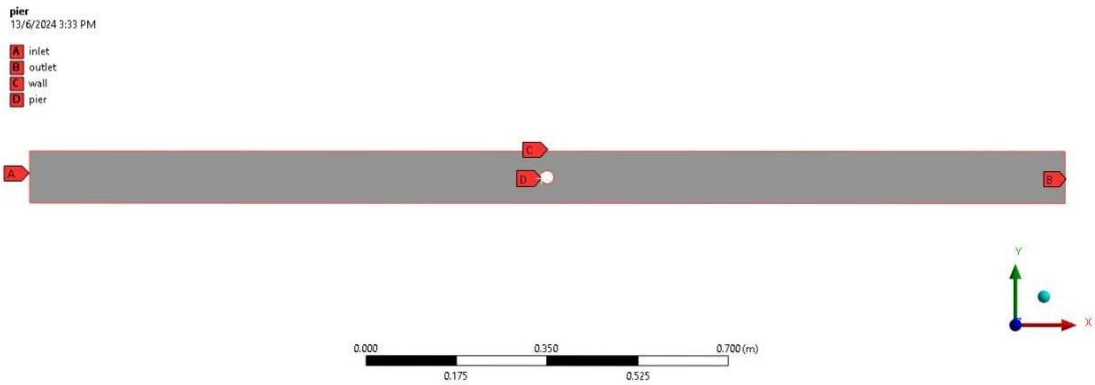

Figure 4. Boundary conditions of circular pier for Case 1

Table 1. Boundary conditions of the piers [21]

Section	Boundary Conditions
Inlet	Velocity Inlet Velocity: 0.041 m/s
	Turbulence Intensity, I: 7.219
	Hydraulic Diameter (4R): 0.0568 m
Outlet	Pressure Outlet
Channel Walls	Wall (Stationary) No slip Condition
	Roughness Height: 0.0 mm
	Roughness Constant: 0.5
Pier	Wall (Stationary)

2.1.1 Barriers at the Upstream of Circular Pier (Case 2)

Vijayasree et al. [19], Prasanna and Kumar [21], and a few other studies have highlighted the effectiveness of using oblong piers in their research to examine water flow around the pier. Ghaderi and Abbasi [15] evaluated the effectiveness of combination countermeasure by simultaneously altering the pier's geometry shape and using collar. Farooq and Ghumman [22] and Singh et al. [6] claimed there would be different impacts when mitigation is applied on the bridge pier. There is the need for flow altering countermeasures which is important as it reduces the strength of horseshoe vertices by changing the flow field. Therefore, barriers also known as fenders were proposed in the current study with a specific angle (30°) and size, instead of using collars or oblong piers. The aim is to investigate and compare the behaviour of water with and without the presence of fenders, in order to assess their effectiveness in protecting the existing pier. Therefore, 2 solid concrete fenders/ barriers were suggested and placed at the upstream of the circular bridge pier. The methods and steps in modelling of the Case 2 are same with the previous one as per Case 1 only the differences would be additional fenders were applied. Figure 5 and Figure 6 show the details of the geometry and mesh generated for Case 2.

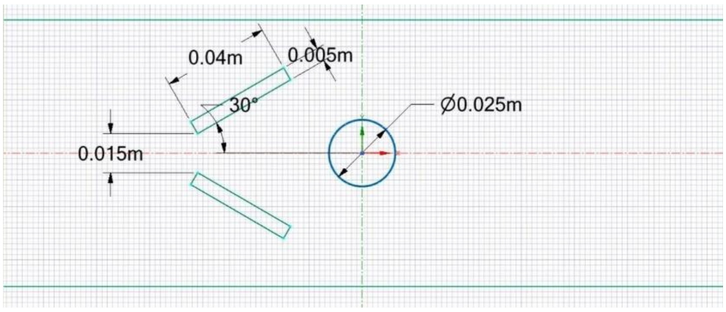

Figure 5. Dimensions of the geometry with barriers/ fenders at the upstream of circular pier for Case 2

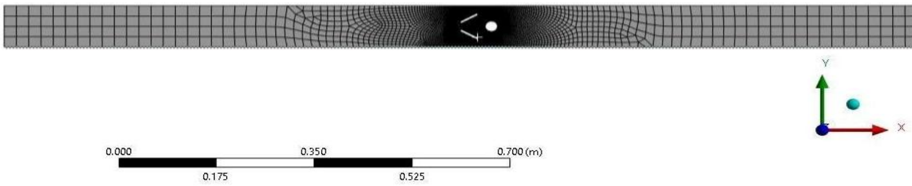

Figure 6. Mesh generated for Case 2

3. RESULTS AND DISCUSSION

Water velocity impact is one of contributors that leads to the bridge pier damages or collapse. Therefore, through numerical analysis on the water velocity behaviours acting on circular bridge pier can be studied mainly velocity contour and contour pressures. Current study validated using the 2-D model designed for Case 1 based on past studies by Prasanna and Kumar [21] ensures that the current methodology is accurately implemented. In order to validate the model analysis result, the modelling of the 2-D model was simulated under the same conditions such as geometry dimensions, meshing size, and boundary conditions. Figure 7 shows the contours of the resulting velocity profile surrounding the pier with the total length of wake region as water flows from left to right in the diagram. Meanwhile, Figure 8 shows the pressure contour around the circular pier. With a mean velocity of 0.041 m/s, the maximum velocity obtained is 0.066 m/s where it is occurring at the sides of the pier as indicated in the red shaded area in Figure 7. Due to the existence of the pier that acts as an obstruction in the water flow, the velocity increases at the sides of the pier as the water flow is split up.

By referring to both Figure 7 and Figure 8, it is clear that the pressure is relatively high at the upstream sides of the pier when the velocity is at rest which is equal to 0 m/s. Due to the presence of the pier, the flow is being disrupted and eventually stops causing the pressure to increase. As the pressure gradually decreases along the downstream sides of the pier, the flow is forced to move downward and may induce erosion at the pier base. This phenomenon is known as scouring which comply with Liang et al [5] findings. From Figure 7, it can be observed that the stagnant point occurs at the downstream side of the pier which causes the formation of a wake vortex. Since the total length of the wake region is 3.76 cm, it depicts that sand will accumulate in the wake region with a total size of 3.76 cm. The formation of the wake vortex is because of the high velocity at sides of the pier that will push and transport the sediments to the downstream. Unfortunately, due to the models computed in 2-D view, the results obtained are unable to predict the total depth of the scour size since flatbed is assumed in current analysis. Table 2 shows the validation results show the percentage difference that is calculated between results in the current study and results from the past study. As the percentage difference achieved is within the requirement which is in good agreement and the results are acceptable. A total of ten positions are chosen to depict the variation of the velocity values. Figure 9 and Figure 10 shows the validation of current study compared with Prasanna and Kumar [21] studies.

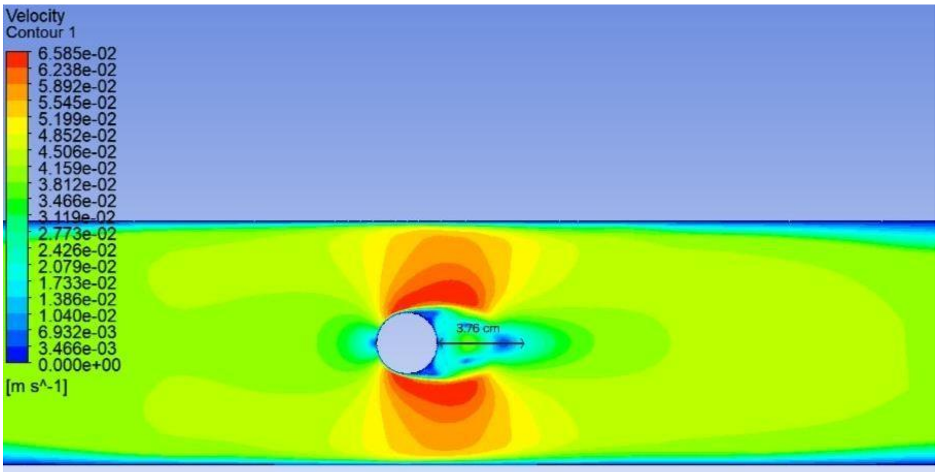

Figure 7. Velocity contours of the circular pier with the total length of wake region of current study

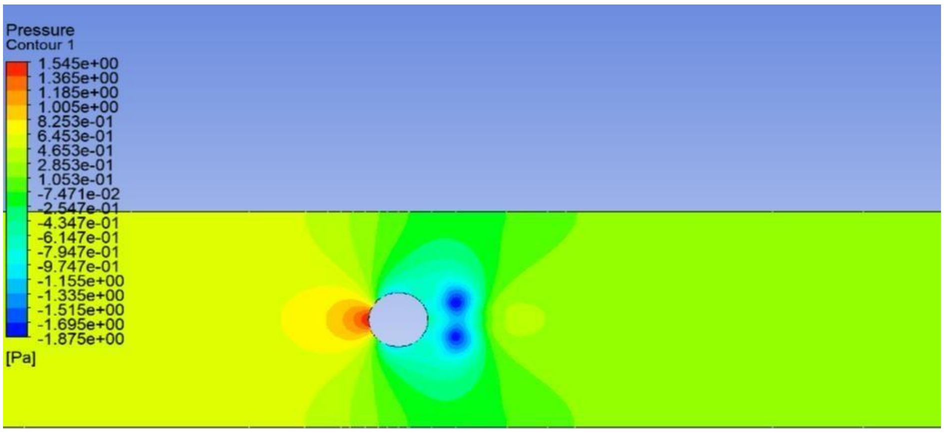

Figure 8. Pressure contours of current study

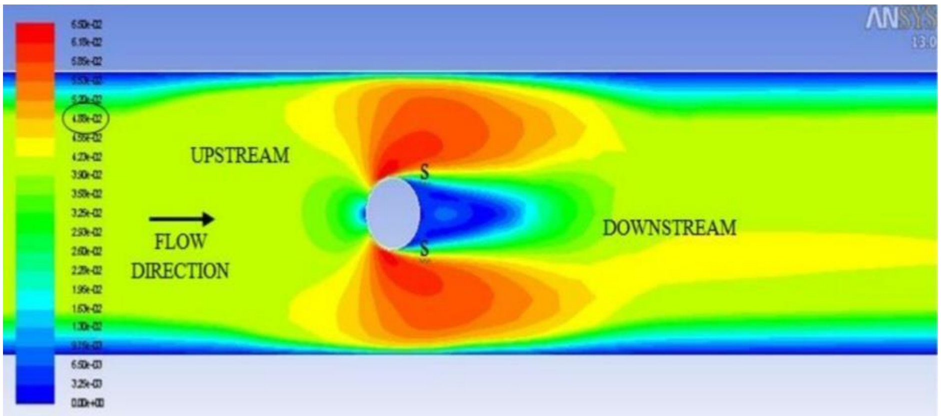

Figure 9. Velocity contours of the circular pier [21]

Table 2. Comparison of results based on Case 1 with past studies

Case 1	Validation Results		Percentage of Differences (%)
	Prasanna and Kumar [21]	Current Study	
Max. velocity (m/s)	0.065	0.066	1.54%
Wake region length (cm)	4.940	3.760	23.89%
Dynamic pressure (Pa)	0.844	0.645	23.58%

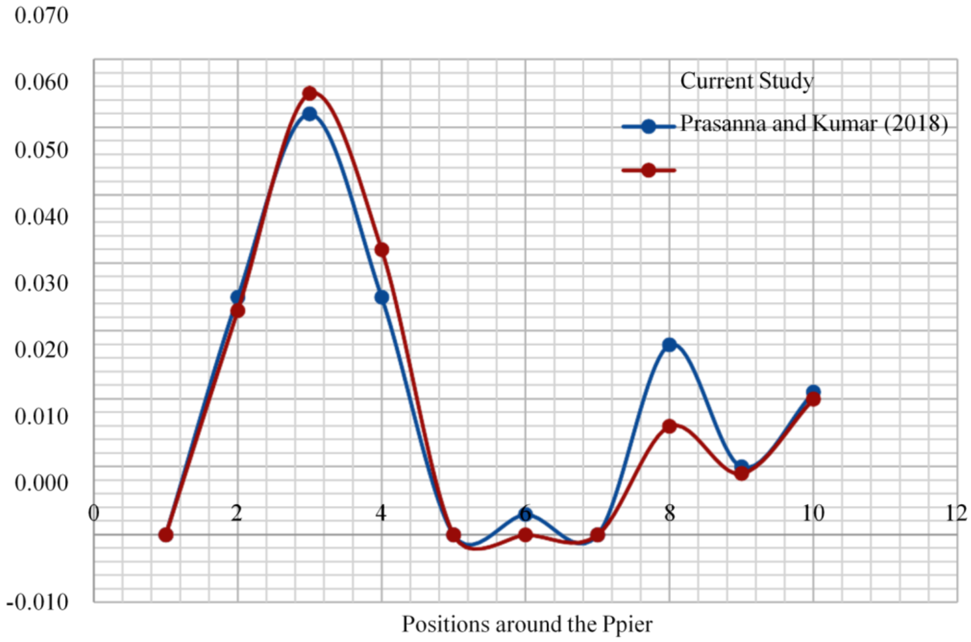

Figure 10. Comparison of velocity (m/s) for validation purposes

For Case 2 study, a simulation of 2-D circular pier with fenders which acts as a barrier is placed at the upstream of the pier. Figure 11 shows the velocity profile surrounding the pier is noticeably different compared to the previous cases (Case 1). For a mean inlet velocity of 0.044 m/s, a maximum velocity of 0.111 m/s that occurs at the side of the pier is obtained. The maximum velocity in Case 2 is much higher than the Case 1 due to the existence of the panels. As stated by Ralston [23] velocity increases when there is a barrier or obstructions along the water flow. Even though the velocity achieved is higher than the first cases, the region with 0 m/s velocity as marked in the blue region is bigger. This indicates that the potential of a scour hole is hard to be formed as the velocity value is too small that is unable to transport the sediment.

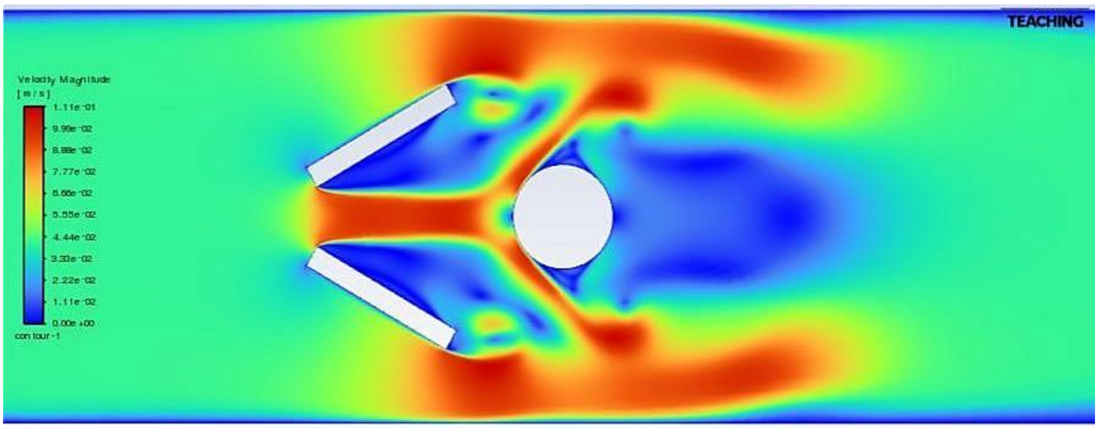

Figure 11. Velocity contour of the second cases

4. CONCLUSIONS

Structure resistance such as bridge pier to flooding is a critical parameter that should be taken into consideration in analysis and design to prevent any structure failures mainly under natural disasters. Moreover, the flood phenomenon has the ability to cause a turbulence flow that leads to increases of water velocity that may impact the bridge piers. The main objective is to investigate and compare the water flow behaviour around a circular bridge pier with and without fender installations, using Computational Fluid Dynamics (CFD). Thus, in the present study, the water velocity behaviour surrounding the circular piers was numerically studied under two different cases. Case 1 is circular pier without fenders and Case 2 circular pier with addition of fenders as barrier. Reynolds-Averaged Navier-Stokes (RANS) implemented in the simulation, as it employs the standard k – ω model to reflect the boundary turbulence conditions. Using ANSYS-Fluent software, the models have been successfully simulated in order to study the water behaviours impact acting on the surrounding circular pier. For Case 1, the water velocity is relatively high at the sides of the pier where the minimum value is about 0.035 m/s. Next for Case 2, when fenders have been placed at the upstream of circular piers, analysis shows that maximum velocity is increased by 68%. Furthermore, the maximum velocity as indicated in the red region is lessened around the pier Case 2. Besides, compared to Case 1, the water velocity impacting the pier at Case 2 is at maximum which indicates a lower pressure distribution to the pier since the relationship between water velocity and water pressure are inversely proportional. The outcomes of analysis discovered that installation of fenders may reduce the water impact at the circular pier that might potentially reduce scour hole formation at the wake region as the velocity obtained is very low. It can be observed that the fenders also acted as barriers to reduce adverse pressure gradient formation, leading to lesser strength of horseshoe vortex of the water flow behaviour. In conclusion, the CFD simulation tool effectively analyses and clarifies the impact of water flow around circular bridge piers, both with and without fenders. In the future, the study recommends incorporating the effects of scouring to gain a deeper understanding of how water impact influences the stability of bridge piers.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the department and IIUM for their continuous support through this project.

FUNDING

This study was not supported by any grants from funding bodies in the public, private, or not-for-profit sectors.

CONFLICT OF INTEREST

The authors declare no conflicts of interest.

AUTHORS CONTRIBUTION

Siti Nur Husna Mohd Yusof: Analysis, Conceptualization,

Tahara Ramadzan Md Kassim: Supervision; Writing; Editing

Nik Zainab Nik Azizan: Original draft preparation; Methodology

Nadiah Md Husain: Data collection; Formal analysis

Nur Khairiyah Basri: Data curation; Reviewing

AVAILABILITY OF DATA AND MATERIALS

The data used to support the findings of this study are included within the article.

REFERENCES

[1] S. A. Mitoulis, S. A. Argyroudis, M. Loli, and B. Imam, "Restoration models for quantifying flood resilience of bridges," *Engineering structures*, vol. 238, p. 112180, 2021.

[2] D. Gautam and Y. Dong, "Multi-hazard vulnerability of structures and lifelines due to the 2015 Gorkha earthquake and 2017 central Nepal flash flood," *Journal of Building Engineering*, vol. 17, pp. 196–201, 2018.

[3] Y. Li, Y. Dong, D. M. Frangopol, and D. Gautam, "Long-term resilience and loss assessment of highway bridges under multiple natural hazards," *Structure and Infrastructure Engineering*, vol. 16, no. 4, pp. 626-641, 2020.

[4] G. A. Holemba and T. Matsumoto, "Flood-induced bridge failures in Papua New Guinea," in *MATEC Web of Conferences*, vol. 258, p. 03014, 2019.

[5] B. Liang, S. Du, X. Pan, and L. Zhang, "Local scour for vertical piles in steady currents: Review of mechanisms, influencing factors and empirical equations," *Journal of Marine Science and Engineering*, vol. 8, no. 1, p. 4, 2019.

[6] N. B. Singh, T. T. Devi, and B. Kumar, "The local scour around bridge piers—a review of remedial techniques," *ISH Journal of Hydraulic Engineering*, vol. 28, sup1, pp. 527-540, 2022.

[7] Florida Department of Transportation, Bridge Scour Manual. [Online]. Available: https://www.fdot.gov/docs/default-source/roadway/drainage/bridgescour/FDOT-Scour-Manual.pdf.

[8] G. Bombar, "Influence of bridge piers on velocity under unsteady flows," *Journal of Innovative Science and Engineering*, vol. 6, no. 2, pp. 279-296, 2022.

[9] C. Huang, C. Tang, and T. Kuo, "Use of surface guide panels as pier scour countermeasures," *International Journal of Sediment Research*, vol. 20, no. 2, pp. 117, 2005.

[10] M. Vaghefi, S. Solati, and C. Abdi Chooplou, "The effect of upstream T-shaped spur dike on reducing the amount of scouring around downstream bridge pier located at a 180° sharp bend," *International Journal of River Basin Management*, vol. 19, no. 3, pp. 307-318, 2021.

[11] J. Ning, G. Li, and S. Li, "Numerical simulation of the influence of spur dikes spacing on local scour and flow," *Applied Sciences*, vol. 9, no. 11, p. 2306, 2019.

[12] G. Misuriya, T. I. Eldho, and B. S. Mazumder, “Turbulent flow field around a cylindrical pier on a gravel bed,” *Journal of Hydraulic Engineering*, vol. 149, no. 10, p. 04023040, 2023.

[13] M. Grioni, S. A. Elaskar, and A. E. Mirasso, “Numerical simulation of flow around circular cylinder near a plane wall: Effects of wall proximity, boundary layer and Reynolds number,” *South Florida Journal of Development*, vol. 4, no. 5, 2023.

[14] P. X. Ramos, R. Maia, L. Schindfessel, T. De Mulder, and J. Pêgo, “Large Eddy Simulation of the water flow around a cylindrical pier mounted in a flat and fixed bed,” in Proc. 6th IAHR Int. Junior Researcher and Engineer Workshop on Hydraulic Structures (IJREWHS), Lübeck, Germany, May 30–Jun. 1, 2016.

[15] A. Ghaderi and S. Abbasi, “CFD simulation of local scouring around airfoil-shaped bridge piers with and without collar,” *Sādhanā*, vol. 44, pp. 1-12, 2019.

[16] J. Wu and C. Shu, “Numerical study of flow characteristics behind a stationary circular cylinder with a flapping plate,” *Physics of Fluids*, vol. 23, no. 7, p. 073603, 2011.

[17] K. M. Sowoud, A. A. Al-Filfily, and B. H. Abed, “Numerical investigation of 2D turbulent flow past a circular cylinder at lower subcritical Reynolds number,” *IOP Conference Series: Materials Science and Engineering*, vol. 881, no. 1, p. 012160, 2020.

[18] S. D. Jasim, M. A. Al-Dabbagh, A. T. Hasen, and A. A. Shekho, “Evaluation of flow behavior around bridge piers,” *Solid State Technology*, vol. 63, no. 6, 2020.

[19] B. Vijayasree, T. Eldho, and B. Mazumder, “Turbulence statistics of flow causing scour around circular and oblong piers,” *Journal of Hydraulic Research*, vol. 58, pp. 673–686, 2020.

[20] L. N. Pasupuleti, P. V. Timbadiya, and P. L. Patel, “Flow field measurements around isolated, staggered, and tandem piers on a rigid bed channel,” *International Journal of Civil Engineering*, pp. 1–18, 2021.

[21] S V S N D L Prasanna and S. N. Kumar, “Simulation of flow behavior around bridge piers using ANSYS–CFD,” *International Journal of Science and Engineering Invention*, vol. 7, pp. 13-22, 2018.

[22] R. Farooq and A. R. Ghumman, “Impact assessment of pier shape and modifications on scouring around bridge pier,” *Water*, vol. 11, no. 9, p. 1761, 2019.

[23] D. K. Ralston, “Impacts of storm surge barriers on drag, mixing, and exchange flow in a partially mixed estuary,” *Journal of Geophysical Research: Oceans*, vol. 127, no. 4, p. e2021JC018246, 2022.