STM JOURNALS

Journal of
Polymer & Composites

ISSN: 2321-2810 (Online)
ISSN: 2321-8525 (Print)
Volume 11, Special Issue 1, 2023
DOI (Journal): 10.37591/JoPC

http://engineeringjournals.stmjournals.in/index.php/JoPC/index

Research JoPC

Flow Structures Around Proposed Bridge Piers

Laxmi Narayana Pasupuleti[1,*], Praful Kumar Vasharam Bhai Timbadiya[2], Prem Lal Patel[3]

Abstract

The current study investigated the flow hydrodynamics around proposed bridge pier and made comparison with existing bridge pier. To quantify the flow structures around the piers, the flow and turbulence parameters are analysed to understand the interference of one bridge pier over another. The experiments were performed on recirculating channel of 15 m length, 0.89 m width and 0.65 m height. Instantaneous three-dimensional velocity data was recorded by acoustic Doppler velocimeter at different radial planes α = 0°, 45°, 90°, 135° and 180° around the piers of both the bridges, where the distance (centre to centre) was maintained two times the diameter of pier (d = 8.8 cm). The flow structures, velocity fields, vector patterns, distributions of turbulence fields are analysed around the piers. The results of planned bridge pier are compared with existing one under identical flow in the clear waters. The results from the current study reveal that the flow velocity was decreased by 30% of the mean flow velocity at α = 180°. Further, the turbulence was significantly reduced around the proposed bridge pier with respect to existing, due to sheltering effect imposed by the front pier. Turbulence intensities, turbulent kinetic energy and Reynolds shear stresses are decreased by 30%, 40% and 30%, respectively. Due to the horseshoe vortex, strength is reduced by 30% around new pier vis-à-vis the old one. The present study recommends that the placement of new bridge pier should be 2 to 3 times the diameter of the pier to avoid more scouring.

Keywords: Proposed bridge, vector fields, turbulence intensity, Reynolds shear stresses, local scour

INTRODUCTION

Flow and turbulence fields around the bridge piers provides insights of horseshoe vortex and its movements. The study of local scour around proposed bridge which is parallel to the existing one is challenging for hydraulic engineers for proper position, alignment, numbers on the river bed [1]. The presence of obstruction in the stream changes the flow pattern, and confined to the small area around the obstruction and is generally associated with three-dimensional flows and vortex system [2, 3]. The flow structures around singe pier and associated turbulence has been studied by many researchers and documented widely [4–9].

Further, Ataie-Ashtiani and Aslani-Kordkandi [10] explored flow characterization around the piers in tandem arrangement along one radial plane, using acoustic Doppler velocimeter (ADV). They concluded that turbulence fields were decreased in tandem case in comparison to single pier. Wang et al. [11] developed an empirical equation to predict the scour depth around twin bridge piers. In addition to this, Keshavarzi et al. [12] explored turbulence fields around tandem circular piers at different spacing using particle image velocimeter (PIV). The study concluded that, if the spacing between the piers was 2 to 3 times of diameter of pier, it creates strong

***Author for Correspondence**
Laxmi Narayana Pasupuleti
E-mail: laxminarayanap@aec.edu.in

[1]Assistant Professor, Department of Civil Engineering, Aditya Engineering College, Surampalem, East-Godavari District, Andhra Pradesh, India, India
[2]Associate Professor, Department of Civil Engineering, Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat, India
[3]Professor, Deputy Director, Department of Civil Engineering, Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat, India

Received Date: December 21, 2022
Accepted Date: March 19, 2023
Published Date: June 19, 2023

Citation: Laxmi Narayana Pasupuleti, Praful Kumar Vasharam Bhai Timbadiya, Prem Lal Patel. Flow Structures Around Proposed Bridge Piers. Journal of Polymer & Composites. 2023; 11(Special Issue 1): S14–S21.

© STM Journals 2023. All Rights Reserved

turbulence behind the front pier. Recently, along with tandem piers, Vijayasree et al. [13] explored flow characterization around circular, two tandem and oblong piers using ADV. The study concluded, turbulence fields are significantly decreased in oblong pier vis-à-vis tandem and single pier.

Extensive experimentations were undertaken on scour around bridge piers to estimate its temporal variations around the piers. Several relationships have been given for the temporal variations of scour depth, and equilibrium scour depth around a single bridge pier from the experimental data. However, turbulence fields around proposed bridge pier in tandem arrangement at different radial planes are not quantified yet to the best of the authors' knowledge. The aim of the present study was to investigate the flow characterization around the parallel bridge piers arranged in tandem configuration and to quantify the effect of flow turbulence at different radial planes (α = 0°, 45°, 90°, 135° and 180°). Due to rapid urbanization, the proposed bridge piers are in need for proper placement, either upstream or downstream to the exiting one.

The limited availability of studies on turbulence characterization around parallel bridge piers are motivated the authors to carry the present study with following objectives:

(a) To identify downflow and corresponding point of boundary layer separation through vector fields.
(b) To compute, turbulence intensity, turbulent kinetic energy, Reynolds shear stresses around both existing and proposed bridge pier and made comparison with existing bridge pier.

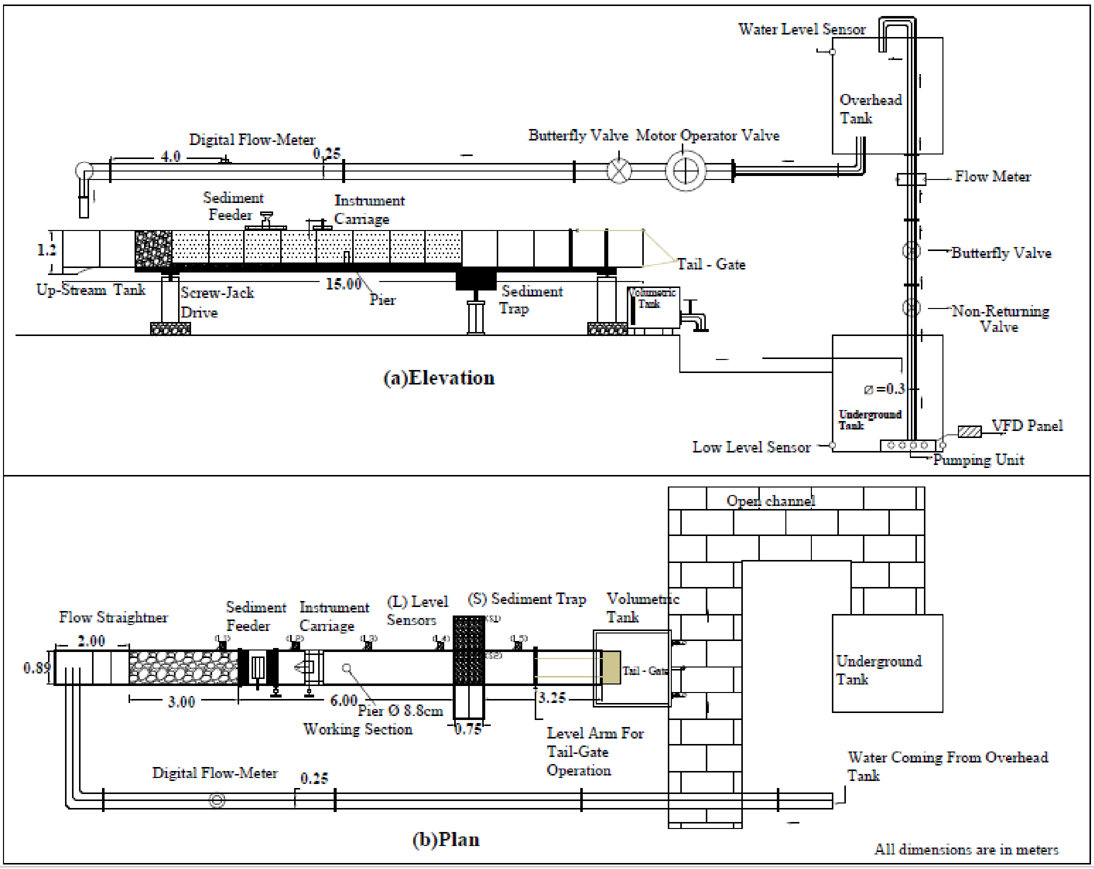

Figure 1. Schematic of experimental set-up: (a) elevation and (b) plan.

EXPERIMENTATION

Experimental set-up

The experimentations were carried in a recirculating channel having dimensions (15 m × 0.89 m × 0.60 m) [14] to fulfil the aforementioned objectives (Figure 1). The flow was released through

Supervisory control and data acquisition (SCADA) system, calibrated by flow meter, inserted into inlet pipe. The discharge given into the SCADA system has been verified through volumetric method using volumetric tank, situated at downstream to tail gate. The uniform flow was kept constant over the 6.0 m working section with the help of tail gate provided at the outlet of the channel.

METHODOLOGY

The flow discharge 0.022 m^3/s was released through SCADA into the channel and predefined uniform depth of 10.5 cm was maintained throughout all the experimental runs. The flow over rigid boundary condition wherein, no sediments are placed over the 6 m working section. However, at most upstream section (3 m long), the bed having 2- to 5-mm size was laid to develop the flow over the working section. A cylindrical pier was placed at 8 m from inlet, along the centreline of channel. To quantify the flow at upstream of pier, the instantaneous velocity data was captured using ADV across the channel at three locations throughout the depth. Here, three dimensional velocities (u, v, w) are expressed in X, Y, Z directions as: $u = \bar{u} + u'$, $v = \bar{v} + v'$, $w = \bar{w} + w'$, where, $\bar{u}$, $\bar{v}$, $\bar{w}$ are time-averaged velocities, and u', v', w' are fluctuating components in X, Y, Z directions respectively. Here, X-, Y-, and Z-axis are considered in longitudinal, transverse, and normal to flow directions respectively. The schematic of pier locations can be seen in Figure 2. The experimental conditions used in the current study are presented in Table 1. After quantification of approaching flow over working section, the velocity data were recorded at different point (see Figure 2). At all the locations, the 7200 data points were recorded with sampling rate of 40 Hz. The noise and spikes among the data had been eliminated using phase space threshold technique [15]. At the end of run-1, the pumps were shut down and allowed the channel to drain. Two cylindrical piers were placed as shown in Figure 2(b) and similar procedure was adopted to collect the data.

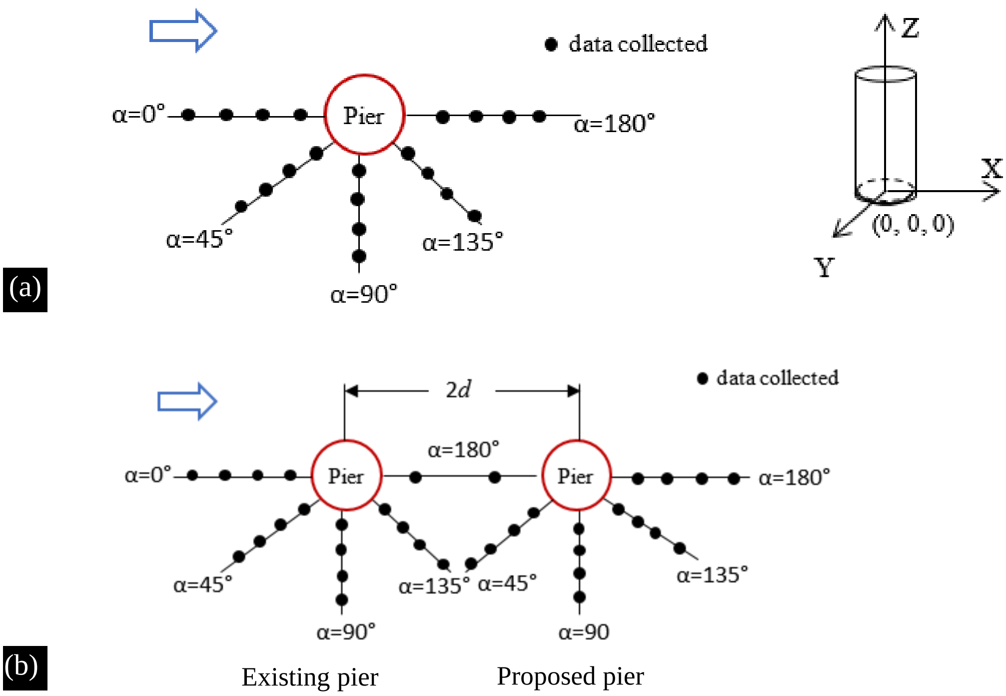

Figure 2. Schematic of velocity data collection around the pier of (a) the existing bridge and (b) the proposed bridge.

Table 1. Flow conditions used in the experiments

Run	Pier arrangements	Discharge (m^3/s)	Flow depth (Z_0) (m)	Flow Reynolds No, Re ($\times 10^4$)	Longitudinal slope ($\times 10^{-4}$)	Mean approaching flow velocity (U) (m/s)
1	Existing	0.022	0.105	2.47	3.8	0.235
2	Proposed	0.022	0.105	2.47	3.8	0.235

© STM Journals 2023. All Rights Reserved

RESULTS AND DISCUSSIONS

Velocity Fields

The processed velocity data were used to compute the velocity fields around both existing and proposed bridge piers to identify the locations of wake zone behind the piers. The contours of velocity fields are shown in Figure 3. Here, Figure 3(a) and (b) represents, the contours of velocity fields at mid flow depths for existing and proposed piers respectively. Figure 3(c) and (d) represents the same at near bed, respectively. From Figure 3, it can be seen that, the velocity was significantly decreased around proposed bridge pier with reference to existing one. It is observed that 30% reduction of approaching average flow velocity around the proposed pier vis-à-vis the existing one. The sheltering effect imposed by old one and reduction of vortex strength over new pier, which can be seen from vector fields.

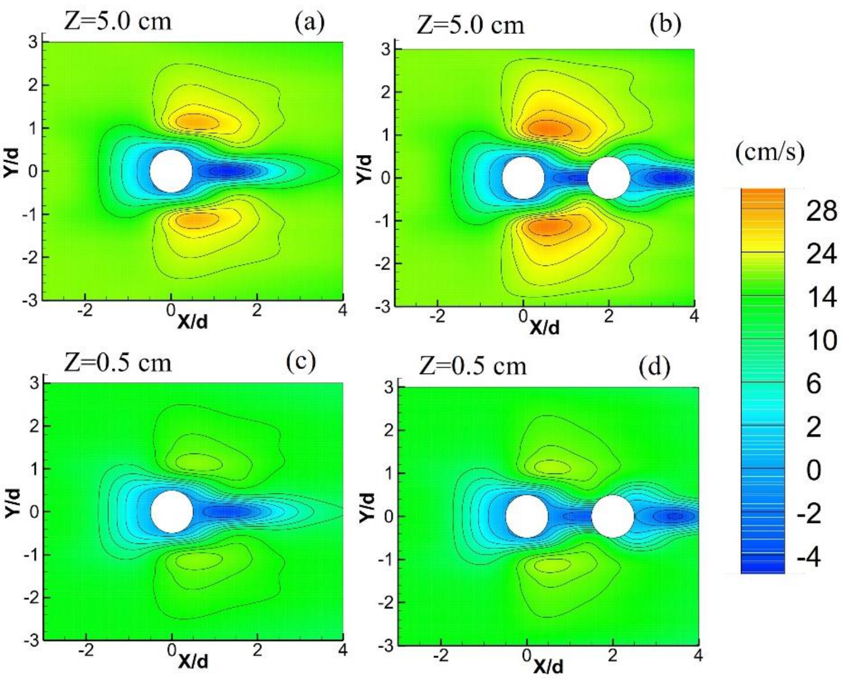

Figure 3. Contours of velocity fields on XY plane over the pier (a, b) at mid flow depth, (c, d) at near bed. (a, c) for the existing bridge and (b, d) for the proposed bridge.

Vector Fields

The vector fields are used to identify the magnitude and direction of instantaneous velocities along and across the flow, which is used to identify the boundary layer separation and point of flow reversal around the piers. In the current study, the vertical distributions along the flow around the existing and proposed bridge pier are shown in Figure 4. Here, Figure 4(a) represents the vector distribution for the existing bridge and Figure 4(b) represents that for the proposed bridge pier. From Figure 4, it can be seen that the point of boundary layer separation for both cases was observed at –1.5*d* and the point of flow reversal was observed at 2*d* and *d* for the existing and proposed bridge piers.

Turbulence Intensities

Turbulence parameters around both the bridge piers were quantified through fluctuating components in three directions as mentioned. The vertical distributions of turbulence intensities are presented around the piers for existing and proposed arrangements are shown in Figure 5. Here, Figure 5(a) and 5(b) represents the distributions of intensities along the flow at different depths for the existing and proposed bridge piers, respectively. In the current study, intensities are normalized by approaching mean flow velocity, U and represented as $\frac{\sqrt{\overline{u'^2+v'^2+w'^2}}}{U}$. Here, $\sqrt{\overline{u'^2}}$ $\sqrt{\overline{v'^2}}$ $\sqrt{\overline{w'^2}}$

© STM Journals 2023. All Rights Reserved

Journal of Polymer & Composites
Volume 11, Special Issue 1
ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)

represent longitudinal (Ti_u), transversal (Ti_v), and vertical (Ti_w) turbulence intensities respectively. From Figure 5, it can be observed that with increase in angle, intensities are increased in both the cases, that is, at $\theta = 180^{\circ}$, the intensities are higher, due to flow separation. The highest intensities are observed at X/*d* = 2 and X/*d* = 1.5 for existing and proposed piers respectively. The evidence can be identifying through critical watch on vector fields wherein, the flow reversal was seen at same locations. Further, the intensities are significantly reduced around proposed pier due to the sheltering effect of front pier and it results in 30% reduction in turbulence intensities.

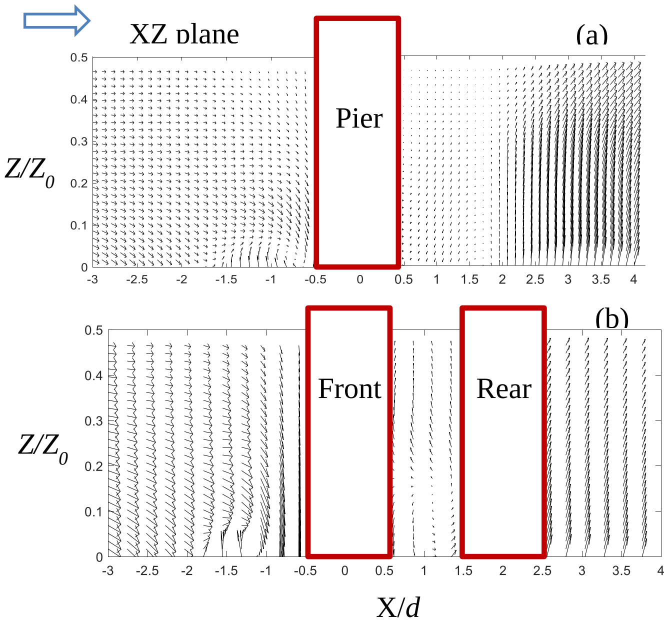

Figure 4. Vector fields on the XZ plane along (a) the existing pier and (b) the proposed pier.

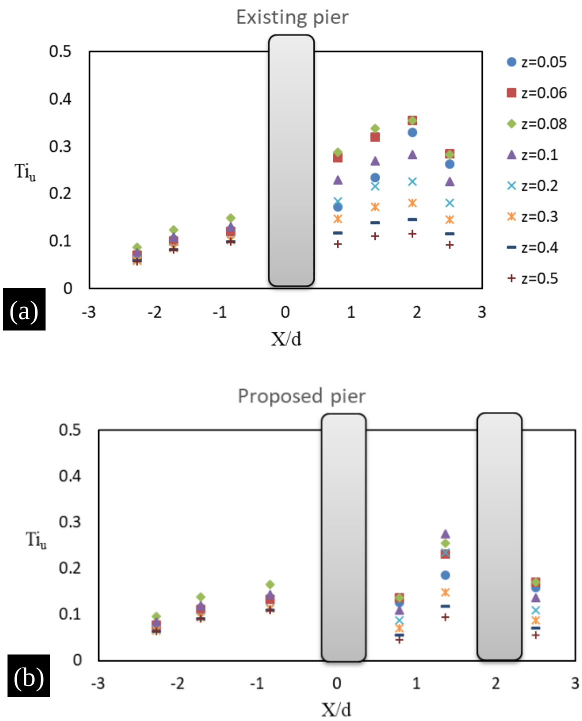

Figure 5. Streamwise turbulence intensities along the XZ plane around (a) the existing pier and (b) the proposed bridge pier.

© STM Journals 2023. All Rights Reserved

Turbulent Kinetic Energy

The vertical distributions of turbulent kinetic energy (k) around both existing and proposed bridge piers are shown in Figure 6(a) and (b), respectively. From Figure 6, it can be seen that k at mid-flow depths are doubled vis-à-vis near the bed in both the cases. Further, the parameter is increased around X/*d* = 2 in both configurations, particularly, existing pier case pronounces, higher energies vis-à-vis proposed case. It is observed that 30% energies are reduced in proposed case. From overall observations, there was a significant reduction in turbulence fields around proposed bridge pier.

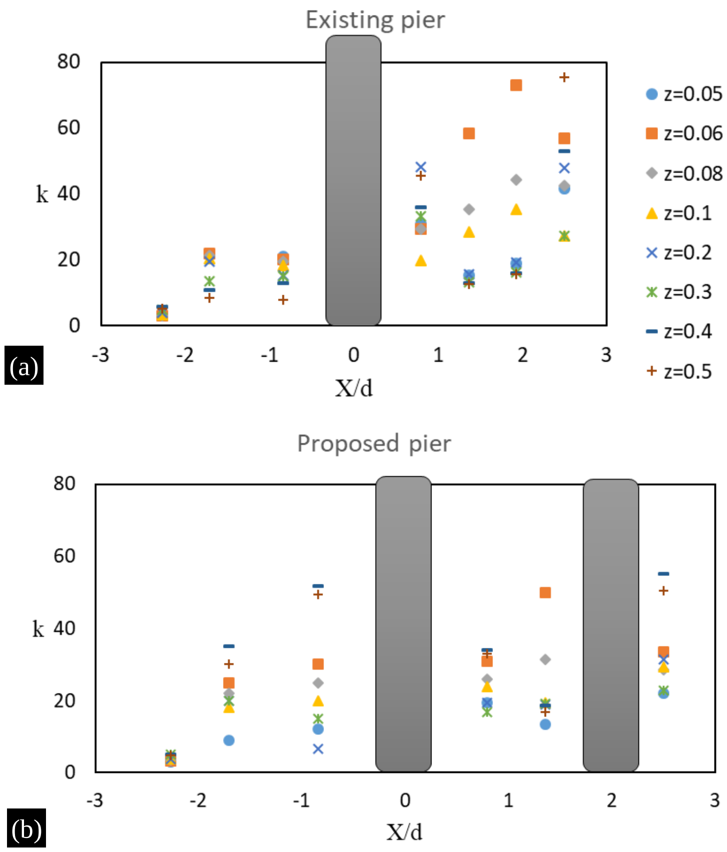

Figure 6. Turbulent kinetic energy distributions along the XZ plane around (a) the existing pier and (b) the proposed bridge pier.

CONCLUSIONS

Turbulence parameters over the proposed downstream pier on rigid bed are investigated in the current study. The measurements of three-dimensional instantaneous velocities using ADV have been used to quantify the vector fields, turbulence intensities and turbulent kinetic energy. The following conclusions are drawn from the current study:

(a) Vector fields reveal that the point of boundary layer separation and point of flow reversal around both the existing and proposed cases. The results from vector fields show that the magnitude of approaching flow velocity has been deceased by 30% at proposed downstream pier.

(b) Due to the flow separation, intensities are shown higher in behind the piers of both cases. Moreover, 30% intensities are decreased at the proposed bridge, due to the sheltering effect imposed by the existing one.

(c) The results of turbulent kinetic energy also shown similar behavior as turbulence intensities. Further, it is seen that the peak values of k at mid-flow depth is around 2.5 times than that near bed. Particularly, the parameter found higher at X/*d* = 2. The presence of the existing pier leads to 40% reduction in energies around the proposed downstream pier.

© *STM Journals 2023. All Rights Reserved*

Hence, the present study recommends that the placement of the new bridge pier should be 2 to 3 times the diameter of pier to avoid more scouring. wherein turbulence parameters are decreased by 30% around new pier vis-à-vis the old one.

Declaration of Interest

The authors declare that there is no conflict of interest regarding the publication of this manuscript.

Acknowledgement

Authors are thankful to Centre of Excellence (CoE) on "Water Resources and Flood Management" of SVNIT Surat funded under TEQIP-II, Ministry of Human Resources Development (MHRD), Government of India, for providing the necessary infrastructure support for conducting the experiments.

Nomenclature

X = streamwise direction

Y = transverse direction

Z = vertical direction

d = diameter of pier (cm)

R_e = flow Reynolds number

U = approaching mean flow velocity (cm/s)

u = instantaneous velocity in X-direction (cm/s)

v = instantaneous velocity in Y-direction (cm/s)

w = instantaneous velocity in Z-direction (cm/s)

$\bar{u}$ = time averaged velocity in X-direction (cm/s)

$\bar{v}$ = time averaged velocity in Y-direction (cm/s)

$\bar{w}$= time averaged velocity in Z-direction (cm/s)

u'= fluctuation velocity component in X-direction (cm/s)

v'= fluctuation velocity component in Y-direction (cm/s)

w' = fluctuation velocity component in Z-direction (cm/s)

Z_0 = approach flow depth (cm)

k = Turbulence kinetic energy (cm^2/s^2)

α = radial planes with respective to flow axis

$\sqrt{\overline{u'^2}}$ = longitudinal turbulence intensity (cm/s)

$\sqrt{\overline{v'^2}}$ = transverse turbulence intensity (cm/s)

$\sqrt{\overline{w'^2}}$ = vertical turbulence intensity (cm/s)

Abbreviations

ADV= acoustic Doppler velocimeter
PIV = particle image velocimeter
SCADA = supervisory control and data acquisition
VFD = variable frequency drive

REFERENCES

1. Kothyari UC, Garde RCJ, Ranga Raju KG. Temporal variation of scour around circular bridge piers. J Hydraul Eng. 1992; 118 (8): 1091–1106.
2. Garde RJ, Raju KG. Mechanics of Sediment Transportation and Alluvial Stream Problems. New Delhi: New Age International Publishers; 2000.

© STM Journals 2023. All Rights Reserved

3. Dey S. Fluvial Hydrodynamics. Berlin: Springer; 2014.
4. Dey S, Bose SK, Sastry GL. Clear water scour at circular piers: a model. J Hydraul Eng. 1995; 121 (12): 869–876.
5. Ahmed F, Rajaratnam N. Flow around bridge piers. J Hydraul Eng. 1998; 124 (3): 288–300.
6. Richardson JE, Panchang VG. Three-dimensional simulation of scour-inducing flow at bridge piers. J Hydraul Eng. 1998; 124 (5): 530–540.
7. Graf WH, Istiarto I. Flow pattern in the scour hole around a cylinder. J Hydraul Res. 2002; 40: 13–20.
8. Muzzammil M, Gangadhariah T. The mean characteristics of horseshoe vortex at a cylindrical pier. J Hydraul Res. 2003; 41 (3): 285–297.
9. Sarkar K, Chakraborty C, Mazumder BS. Variations of bed elevations due to turbulence around submerged cylinder in sand beds. Environ Fluid Mech. 2016; 16 (3): 659–693.
10. Ataie-Ashtiani B, Aslani-Kordkandi A. Flow field around single and tandem piers. Flow Turbulence Combustion. 2013; 90 (3): 471–490.
11. Wang H, Tang H, Liu Q, Wang Y. Local scouring around twin bridge piers in open-channel flows. J Hydraul Eng. 2016; 142 (9): 06016008.
12. Keshavarzi A, Ball J, Khabbaz H, Shrestha CK, Zahedani MR. Experimental study of flow structure around two in-line bridge piers. Proc Inst Civil Eng Water Manage. 2018; 171 (6): 311–327.
13. Vijayasree BA, Eldho TI, Mazumder BS. Turbulence statistics of flow causing scour around circular and oblong piers. J Hydraul Res. 2019; 58 (4): 673–686.
14. Pasupuleti LN, Timbadiya PV, Patel PL. Flow field measurements around isolated, staggered, and tandem piers on a rigid bed channel. Int J Civil Eng. 2022: 20: 569–586.
15. Pasupuleti LN, Timbadiya PV, Patel PL. Bed level variations around submerged tandem bridge piers in sand beds. ISH J Hydraul Eng. 2020; 28 (1): 149–157.

© STM Journals 2023. All Rights Reserved