Large eddy simulation of flow around semi-conical piers vertically mounted on the bed

Yasin Aghaee-Shalmani · Habib Hakimzadeh

Received: 7 July 2021 / Accepted: 27 June 2022 / Published online: 28 July 2022
© The Author(s), under exclusive licence to Springer Nature B.V. 2022

Abstract
In this paper, details, and results of three-dimensional numerical modeling of flow around the semi-conical piers vertically mounted on the bed in a channel, are presented. For flow simulation, 3-D Navier–Stokes equations are solved numerically using the finite volume method and large eddy simulation. In this study, the semi-conical piers with different side slope angles (α) are tested, and the flow around them is compared with the cylindrical reference pier. Flow structures, vortex shedding behind piers, horseshoe vortices, instantaneous and time-averaged flow structures are presented and discussed. Numerical model results show that the semi-conical piers are eventuated remarkable reduction (up to 25%) in downward flow velocity in the upstream side of the piers, and much more reduction (up to 46%) in bed shear stresses in comparison with the cylindrical pier. Moreover, the model results showed some decrease in vortex shedding frequency for the semi-conical piers compared to the cylindrical pier.

Article highlights
We report on numerical results of large eddy simulation of the flow around semi-conical piers with different side slopes. This research is significant because of the effect of these piers on the:

• Reduction of the downward flow and the bed shear stress around the piers.
• Reduction of vortex shedding frequency for the semi-conical piers compared to the cylinder.
• Different behavior of the horseshoe vortices at the upstream compared to the cylinder.

Keywords
Semi-conical piers · Horseshoe vortex · Flow · Numerical model · LES

Yasin Aghaee-Shalmani
y.aghaee@iaukhsh.ac.ir

1 Department of Mechanical and Civil Engineering, Khomeinishahr Branch, Islamic Azad University, Khomeinishahr, Iran

2 Department of Civil Engineering, Sahand University of Technology, Tabriz, Iran
List of symbols

$C_D$ Drag force coefficient (–)
$C_L$ Lift force coefficient (–)
$C_p$ Pressure coefficient (–)
$C_s$ Smagorinsky constant parameter (–)
$D$ Mean diameter of the pier (m)
$D_1$ Diameter of the semi-conical pier at free-surface level (m)
$D_2$ Diameter of the semi-conical pier at bed level (m)
$F$ Froude number (–)
$f$ Frequency of vortex shedding (Hz)
$F_D$ Drag force (N)
$F_L$ Lift force (N)
$G$ Filter function
$g$ Gravity acceleration (ms$^{-2}$)
$h$ Flow depth (m)
$l_s$ Sub-grid length scale (m)
$p$ Filtered pressure (Nm$^{-2}$)
$p_0$ Static reference pressure (Nm$^{-2}$)
$R$ Reynolds number (–)
$R_D$ Reynolds number based on the mean diameter of the pier (–)
$R_{D_2}$ Reynolds number based on the biggest diameter of the pier (–)
$S_{ij}$ Strain rate tensor for the resolved scale (s$^{-1}$)
$St_D$ Strouhal number based on the mean diameter of the pier (–)
$St_{D_2}$ Strouhal number based on the biggest diameter of the pier (–)
$t$ Physical time (s)
$u_i$ Filtered velocity (ms$^{-1}$)
$u_0$ Inlet velocity (ms$^{-1}$)
$u_*$ Bed shear velocity (ms$^{-1}$)
$y$ Distance to the nearest wall (m)
$z$ Distance of the center of the first of wall computational cell (m)
$z^+$ Non-dimensional distance from the wall based on the shear velocity (–)
$\alpha$ Side slope angle of semi conical pier (–)
$\beta$ Compressibility parameter (ms$^{-1}$)
$\Delta$ Volume of the computational cell (m$^3$)
$\delta_{ij}$ Kronecker delta (–)
$\kappa$ Vonkarman constant (–)
$\nu$ Kinematic eddy-viscosity of fluid (m$^2$s$^{-1}$)
$\nu_t$ Sub-grid eddy-viscosity (m$^2$s$^{-1}$)
$\rho$ Density of fluid (Kg m$^{-3}$)
$\tau$ Virtual time (s)
$\tau_{ij}$ Sub-grid shear stress (Nm$^{-2}$)
$\tau_m$ Reference bed sher stress (Nm$^{-2}$)
$<.>\text{ Tim}e\text{-averaged quantity (–)}$
1 Introduction

Numerous researchers have already investigated the flow structures and scour process around piers experimentally [6, 11, 12, 14, 53]. Flow around cylinders includes downflow and horseshoe vortex (HV) infront of the pier, and vortex shedding of wakes at the lee side. The wakes and the frequency of the vortex shedding depend on the Reynolds number of the pier. The wake vortices behind the piers are not exactly similar to those for very long cylinders and piers mounted on the bed, even though the Reynolds number is the same [26, 30]. At the location of the HV, the increase in the bed shear stress and turbulence intensity is evident [11, 21]. HV is mainly generated by downflow and separation of the flow upstream of the pier, and it can be classified as laminar or turbulent based on the Reynolds number of the pier and the characteristics of the incoming boundary layer. Paik et al. [42], Devenport and Simpson [13] showed that the HV exhibits bimodal oscillation behavior for turbulent HV. The turbulence intensity and the increased bed shear stress within the HV system carry the sediment particles and develop the local scour around the pier [19].

The shape and the size of the pier have substantial impacts on the size and strength of the HVs. Regarding the form of the pier, one of the parameters that affect the HVs is the lateral slopes of the pier. The effect of this parameter can also be seen in scour depth for such piers. [7, 54] reported reducing scour depth for inclined pier toward the downstream. [16, 44, 52], and [4] in their experimental studies have shown that the conical or semi-conical piers can reduce the scour depth. Sumer et al. [52] indicated that the bed shear stresses around cone-shaped piers are decreased as the $\alpha$ is increased, and robustness of the HVs are mainly reduced compared to a circular cylinder.

There are a few studies about flow around cone-shaped obstacle mounted on the bed. Among them [41], in a numerical and experimental research, studied the flow around the conical island with a large side slope in shallow water conditions and the flow structures around the cone, including the recirculation region, vortex shedding, and separated shear layers were discussed. Existent experimental studies of flow around semi-conical piers relevant to investigating the flow structures and vortex shedding behind them are almost restricted to the very tall and slim cylinders without interactions from the walls [24, 25]. These experiments are relevant to aircraft or missile industrial investigations [25]. The vortex shedding with oblique and cellular patterns and frequency of wakes have been studied in the mentioned researches with large tapering ratios. [43] reported the cellular behavior of the wake vortices behind tapered cylinders. Also, oblique vortex shedding of wakes for tapered plate [39] and for thin and tall tapered cylinders [43, 55]. Williamson [56] reported some orientation for tall cylinders in wakes. For the tall cylinders, the axes of the vortex tubes are nearly parallel to the cylinder’s axis. However, the presence of the free-surface or bed interaction may cause some oblique vortex shedding [57].

The experimental investigations of the flow around hydraulic structures are somewhat costly and sometimes inaccessible. Therefore, numerical modeling can be a good alternative to experiments. Most of the available flow numerical simulations and scour around piers have been carried out using Reynolds averaged Navier–Stokes equations models (RANS) [2, 38, 40, 49]. RANS methods compared to the large eddy simulation(LES) method are not successful enough to predict some of the flow characteristics around bluff bodies. [3, 9, 46]. Also, direct numerical simulation (DNS) of turbulent flows is too expensive and time-consuming [47]. Therefore, LES is a reliable and efficient method to solve the turbulent flows at moderate or high Reynolds numbers.
in a numerical study, modeled the flow around the cylinder on a scoured bed using LES. Also, [32, 42] simulated the turbulent flow around a pier at high Reynolds numbers using detached eddy simulation (DES). In low and medium Reynolds numbers, the LES method is possible without wall function and with the appropriate number of grids [31, 32]. [29] used LES to investigate the flow around vertical cylinders and the laminar HVs around them. Effects of various Reynolds numbers (between low and high) on the flow structure, turbulence, and behavior of HVs has been investigated comprehensively with particle image velocimetry (PIV), and DES by [30]. Zhang et al. [58] developed a finite volume code for modeling the flow and scour around the cylinder using LES, and the form and strength of the HV during the scour process were explained.

To the best of our knowledge, there is not any detailed numerical study about the turbulent flow around semi-conical (tapered) piers mounted on the bed. In this study, the turbulent flow around semi-conical piers vertically mounted on the bed has been studied numerically using LES. Governing equations were numerically solved using the finite volume and artificial compressibility methods. To solve the equations in time, the dual-time stepping method was used. The flow regime in the channel was subcritical, and the Reynolds number of the flow based on the free stream velocity, pier diameter, and fluid viscosity was turbulent. The bed was assumed fixed and straight with no sediment movements. The rigid-lid assumption was applied for water free-surface.

The focus of this study is on the flow structure around tapered piers with different side slopes. Since the HVs and wakes behind piers are important in the scour process, the bed shear stresses and frequency of vortex shedding, and the properties of the HV are discussed. The importance of the present study is because of the shortage of data about flow structure around semi-conical piers mounted on the bed. This study may reveal some auxiliary information on this issue.

2 Numerical method

In this work, the governing equations of flow are Navier–Stokes equations. The equations were discretized using the finite volume method, and LES was adopted to simulate turbulent flow using the Smagorinsky model. The artificial compressibility method (AC) with dual-time stepping was used to solve the discretized unsteady flow equations [8, 18, 28]. Equations for large-eddy simulation are derived by filtering the Navier–Stokes equations in space. The filtering operation is as follows:

$$\tilde{\phi} = \int \phi(x')G(x - x')dx',$$

(1)

where, $G$ is a filter function. A finite-volume discretization involves a top-hat (box) filter. Herein, The size of the filter is the cubic root of the cell volume. The form of the filtered Navier–Stokes equations based on the LES and dual-time stepping AC is as follows:

$$\frac{1}{\rho \beta^2} \frac{\partial \bar{p}}{\partial \tau} + \frac{\partial \bar{u}_i}{\partial x_i} = 0,$$

(2)

$$\frac{\partial \bar{u}_i}{\partial \tau} + \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_i x_j} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j},$$

(3)
Where $x_i$ are the Cartesian coordinates, $\rho$ is the fluid density, $p$ is the pressure, and $\nu$ is the kinematic viscosity of the fluid. $\tau_{ij}$ is the sub-grid turbulence stress, and the overbar shows the spatial filtering; $t$ is the real time, and $\tau$ is the artificial (pseudo) time. In the dual-time stepping method, instead of solving each time step in real time, the problem is transformed into a sequence of steady-state computations in artificial times. $\beta$ is the artificial compressibility parameter, and its value was adopted equal to inlet flow velocity in the computational domain [3, 5].

Filtering operation separates the solution to resolved large scales and sub-grid scale, which is unresolved and must be modeled. The simplest model for modeling the sub-grid scales of turbulence (Smagorinsky) was used in this study. $\tau_{ij}$ represents the contribution from the sub-grid scales and must be modeled to close the system of equations. The eddy viscosity concept for the sub-grid stress is:

$$\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = -2 \nu_t \bar{S}_{ij}, \quad \nu_t = l_s^2 |\bar{S}_{ij}|,$$

(4)

where $\nu_t$ is the sub-grid eddy-viscosity coefficient, and $\delta_{ij}$ is the Kronecker delta. $\bar{S}_{ij}$ is the strain rate tensor for the resolved scale and $l_s$ is the sub-grid length scale. $l_s$ depends on the grid size and Smagorinsky constant parameter ($C_s$),

$$l_s = \min(\kappa \gamma, C_s \Delta^\frac{1}{3}),$$

(5)

$\Delta$ is the volume of the computational cell, $\kappa$ is the Von Karman constant, and $\gamma$ is the distance to the nearest wall. $C_s$ is the sub-grid scale parameter. The value of $C_s$ depends on the problem. However, the large values of $C_s$ cause excessive damping of the large-scale fluctuations, and very small values ($C_s < 0.1$) may cause convergence problems [35]. The optimum values of the $C_s$ can be found in the literature as $0.1 < C_s < 0.14$. In this research, the parameter $C_s$ was adopted as 0.11.

The momentum interpolation method (MIM) used by [10] was applied to overcome the pressure oscillation due to the collocated grid. Time discretizations of the equations were carried out using the implicit first-order forward Euler method for virtual time and backward second order for real-time. Convective terms were discretized using the QUICK scheme. The differed correction method of [27] was used to deal with convective terms in implicit discretization. The implicit part of the convective terms was linearized by the Newton method [51]. Viscous terms were discretized using the second-order central scheme, and the pressure gradient was approximated by the central scheme. The implicit point Gauss-Seidel method with under relaxation was then applied to solve the linearized system with a seven-diagonal matrix. The values of the physical time step ($\Delta t$) and artificial time step ($\Delta \tau$) depend on the discretization scheme. This backward differencing scheme is strongly stable and dissipative, and thus it is nearly insensitive to the stiffness of the problem [28]. The decomposition of the computational domain to equally spaced partitions was carried out to equally divide the computational cost between processors using the message passing interface method (MPI) to transfer data between the processors. For every run, 16 to 20 processors were used in the simulations. Because of using the QUICK method for convective terms and the momentum interpolation method, it was needed to pass the data of two layers of the decomposed domains between two adjacent domains (processors). The messages were sent using non-blocking send and receive MPI functions. The solver code has been applied to internal and external flows like cavity flow and flow around a vertical cylinder [3, 5].
2.1 Computational domain and boundary conditions

In this study, the flow past semi-conical piers vertically mounted on the bed was simulated based on the configurations in the experimental study of scour around semi-conical piers by [4] in which only scour depth with no flow measurement were taken.

Table 1 presents the detail of the models and flows conditions. The first row of the table shows the details of the conical island model with $\alpha = 35^\circ$ which was used for verification of the numerical model, where $\alpha$ is the inclination angle of the sides with the vertical axis. The subsequent rows are the details of the semi-conical piers. In this table, $D_1$ and $D_2$ are the upper and lower base diameters of the semi-conical pier at the free-surface and bed levels, respectively. Moreover, $D$ is the mean diameter of the models ($D = (D_1 + D_2)/2$) (see Fig. 1). For the semi-conical piers the flow depth was $h = 2.5D$ for all of the tests. The $\alpha$ of the semi-conical piers which ($\alpha = \tan^{-1} (D_2 - D_1)/2h$), are equal to $0^\circ$, $5^\circ$, $10^\circ$, and $15^\circ$, and for the conical island is $35^\circ$. The Reynolds number based on the mean diameter of semi-conical piers ($R_D$) was 7,600 and the Reynolds number of base diameter $D_2$ ($R_{D_2}$) was 7,600 to 12,690 from $\alpha = 0^\circ$ to $\alpha = 15^\circ$. For all of the semi-conical piers, the projected area perpendicular to the flow direction was kept constant. A schematic view of the numerical domain and a semi-conical pier are depicted in Fig. 1. The length of the computational domain was 20D. The computational domain was discretized by an O-type mesh, created by an elliptic Poisson differential equation to adjust the grid size contraction around the piers. The O-type mesh is a usual type of mesh for modeling flow around cylinders or bluff

| Side slope $\alpha$ | $D$ (m) | $D_1$ (m) | $D_2$ (m) | $h$ (m) | Flow velocity (m$^{-1}$) | $R_D$ | $R_{D_2}$ | $F$ |
|---------------------|---------|----------|----------|--------|-------------------------|-------|----------|-----|
| 35$^a$              | 0.081   | 0.0071   | 0.1625   | 0.11   | 0.1                     | 8125  | 16250    | 0.0962 |
| 0                   | 0.04    | 0.04     | 0.04     | 0.1    | 0.19                    | 7600  | 7600     | 0.1918 |
| 5                   | 0.04    | 0.03125  | 0.04875  | 0.1    | 0.19                    | 7600  | 9260     | 0.1918 |
| 10                  | 0.04    | 0.02237  | 0.05763  | 0.1    | 0.19                    | 7600  | 10950    | 0.1918 |
| 15                  | 0.04    | 0.01312  | 0.06679  | 0.1    | 0.19                    | 7600  | 12690    | 0.1918 |

$^a$The first row relates to the specifications of the experimental model of [41], and the rest are for the numerical models of the present study.
bodies [22, 33, 34]. Also, the clustering of the grids was carried out near the bed with a hyperbolic mesh generator. The computational domain length in the present study was long enough at the upstream of the pier to allow the current to become fully developed. The logarithmic velocity distribution of the flow at the inlet ensures that the flow is fully developed over a short length at the upstream of the piers. Different grids were tested for simulation, and $160 \times 160$ for the horizontal plane, and 64 grids were chosen for flow depth. A logarithmic flow distribution was applied at the inlet, and the convective boundary condition at the outlet was used to ensure no reflection from the outlet.

The inlet boundary was located at the left half of the domain, and the outlet was located at the right half. The mean flow velocity at the inlet for semi-conical models was about 0.19 ms$^{-1}$, and the flow depth was set to be 0.1 m. The boundary condition at the bed and the pier’s surface was the no-slip condition, and at the top of the flow, the rigid-lid assumption was applied. The rigid lid assumption for the free surface at low Froude number ($F < 0.2$) is acceptable due to small bow wave height in front of the pier [29, 32, 48]. For assessing this claim, the flow with free-surface and rigid-lid assumptions across the semi-conical piers ($\alpha = 0^\circ$ and $10^\circ$) were tested. No meaning differences were found in comparisons (the numerical results are not shown here for brevity).

Based on the grids in the present study, the smallest grids size near the bed had a non-dimensional length of about $z/D = 1.06 \times 10^{-3}$, equivalent to $z^+ = u_* z / \nu = 1 \sim 3$. Where $u_*$ is the bed shear velocity, $z$ is the distance of the center of the first off-wall computational cell from the rigid wall, and $\nu$ is the kinematic viscosity of the fluid. It seems that this resolution for grids is adequate for LES modeling. The maximum grid size in the flow depth was about 25 wall units close to the free surface.

2.2 Verification model

The experimental flow measurements around the conical pier vertically mounted on the bed are rare. [41] is the only similar study on this issue that experimentally measured the flow field around cone island. In the present study, the numerical model was applied to compute the configuration of [41] for verification purpose and then applied to the conical piers with the smaller $\alpha$. Details of the cone island configuration [41] are given in the first row of Table 1. For cone island, $\alpha$ is $35^\circ$ to the vertical axes and the base diameter $D_2 = 0.1625$ m, with a flow depth $h = 0.11$ m. The average cone diameter was $D = 0.081$ m, which was introduced with $D_{0.5}$ by [41] and is equivalent to $D$ in the current study. The mean velocity at the inlet of the channel for the conical island was 0.1 ms$^{-1}$, and the flow rate was equal to 0.0133 m$^3$s$^{-1}$. For cone island, the relative depth was equal to $h/D = 0.677$. The computational mesh and boundary conditions were the same as the semi-conical piers. The cone was placed at the center of the computational domain. The value of $F$ was 0.1. Due to small $F$, the influence of free-surface effects can be neglected.

3 Numerical model results

3.1 Validation of numerical code, time-averaged velocity field

Flow around obstacles (piers) mounted on the bed is entirely three-dimensional and complex, especially near the up- or downstream side of them. There are significant temporal variations in vortices and the structure of 3D flow and HV. The time-averaging process is
needed to compare the results with each other, along with the most temporal variations in the flow structure. For the present study, time-averaged quantities are obtained for about 20 complete vortex shedding (Karman vortex). This length of time is adequate for time-averaging in LES of flow around cylinders [8]. For validation of the numerical model, the time-averaging process has been carried out for about 160 s for the cone island. This value was adopted similar to the experiment of [41]. Figure 2 depicts the time-averaged streamlines at the vertical plane of symmetry up- and downstream of the cone island for the current LES study (top), experimental results (middle), and LES results (bottom) of [41]. The results were obtained using time-averaged streamwise and vertical velocities normalized with $u_0$. As can be seen from the figure, the experimental results of [41] were only presented for $z/D < 0.9$. The figure for the present study shows the deflection of streamlines at the upside of the cone and recirculation behind it. The deflection occurs at the height of $z/D \approx 0.3$, where the flow tends to move upward above this height and downward below this height. This downward flow causes the HV, as can be observed in the figure.

![Figure 2](image-url)

**Fig. 2** Comparison between numerical results of LES for this study and the experimental and numerical results of [41]. Streamlines and contour of the normalized time-averaged streamwise velocity ($< u > / u_0$) at vertical plane of $y/D = 0$. Upper figure is the LES of present study, the middle is graphical data of experiment and the lower figure is the LES of [41]. (The middle and lower figures reproduced from [Large-eddy simulation of shallow turbulent wakes behind a conical island", Phys. Fluids 29, 126601 (2017)], with the permission of AIP Publishing)
The location of deflection in the present study is similar to LES results of [41]. It should be noted that the downward motion below \( z/D = 0.3 \) was not measured by [41]. Therefore, doubts exist in this location.

In the numerical results of this study, the main recirculation can be observed downstream of the cone. The length of the recirculation near the free surface is greater than near the bed. The largest longitudinal extent of the recirculation is about \( x/D = 2.3 \) which is slightly smaller than the LES of [41]. The overall shape of the recirculation behind the cone in the present study is identical to the experiment. The reattachment position near the bed is located at \( x/D = 1.65 \), \( x/D = 1.8 \), and \( x/D = 2.2 \) for the LES of the present study, experiment, and LES of [41], respectively.

Figure 3 shows the time-averaged streamlines at the horizontal plane at \( z/D = 0.37 \). In this figure, the contours of streamwise velocity (Fig. 3a) and vertical velocity (Fig. 3b) are presented. Experimental data is represented in the upper half of the figure, and the lower half is for the present study. As can be observed from this figure, the recirculating length and contours for the experiment and LES are the same. The shapes of the streamlines behind the cone are slightly different. In Fig. 3b the contours of normalized vertical velocities for both experiments and LES are very similar, except for the upstream of the cone, where the LES shows greater upward velocities. This red region can also be found in the LES of [41].

The profiles of time-averaged normalized streamwise velocities and vertical velocities at the downstream of the cone at locations \( x/D = 1.2, 2.5 \) and 4.3 are presented in Fig. 4. The lines are for streamwise velocities, and the dashed lines are for vertical velocities. The circles and triangles belong to experimental data of [41]. The normalized time-averaged vertical velocities at all locations show very good agreement with experimental data. The profiles of normalized time-averaged streamwise velocities have good agreement with the experiment, except for the data near the bed. This discrepancy could be attributed to acoustic doppler velocimeter (ADV) errors near the solid boundaries of the bed and cone.

Time-averaged streamlines and velocities, as well as vertical profiles of them at the plane of symmetry \( y/D = 0 \), show excellent agreement with experiments. Based on the comparison of the results of the present study with experimental data, it can be claimed that the numerical model of the present study has a very good ability to model the flow around obstacles, particularly around conical objects. Therefore, it can be expected that

![Fig. 3 Streamlines of time-averaged normalized velocities at horizontal plane z/D = 0.37, the upper half of figure is for experiment of [41] and lower half is for present study](image-url)
the application of numerical code to model the flow around semi-conical piers will also provide reliable results.

### 3.2 Semi-conical piers, time-averaged values

Figure 5 shows the time-averaged streamlines velocities at the upstream and downstream sides of the semi-conical piers at the vertical plane of flow. At the upstream of the piers, the streamlines are similar, and a small corner vortex (CV) is observed close to the upside of the pier. There are no secondary HVs in the time-averaged flow due to the low Reynolds number of the flow around the pier. It should be noted that multiple HVs are present at the upstream of the piers in the instantaneous results, which are wiped out in the time-averaging process.

As can be observed from Fig. 5, the separation point (saddle point) at the upstream of the pier near the bed gets closer to the upside of the pier with the increase of the side slope angles. The main flow recirculation at the downstream of the cylinder rotates in the anti-clockwise direction, and the second one rotates near the bed in a clockwise rotation. For the semi-conical piers at the downstream, the vortices near the bed are smaller than the cylinder. The recirculation length in the wake region is about $1.3D_2 \sim 1.5D_2$ (distance from the center of the piers). These values are consistent with findings in researches of flow around the cylinder [8].

As mentioned in the numerical modeling of flow around a cone, the flow upstream of the cone at a height from the bed was deflected into the upward and downward flow. This location depends on the side slope of the cone. As shown in Fig. 5, the height of this deflection decreases as the $\alpha$ of semi-conical piers increases. The time-averaged values of the vertical velocities ($<w>$) at the up-side of the pier have been compared in Fig. 6. As can

Fig. 4 Time-averaged profile of normalized streamwise and vertical velocities at vertical planes at $x/D = 1.2$, $2.5$ and $4.3$ downstream of the cone. Comparison between present numerical model results and experimental data of [41]
be seen from this figure, the magnitude of the down-flow velocity near the bed is gradually decreased by increasing the side slope angles. This reduction is about 25% for the semi-conical pier with $\alpha = 15^\circ$ compared to the cylinder. The location of flow deflection from

---

**Fig. 5** Time-averaged streamlines using $u$ and $w$ velocities at the up and downstream of the semi-conical piers at the plane of symmetry $y/D_2 = 0$ (flow is from left to right)
the bed at the upside of the piers is $z/h = 0.81, 0.7, 0.4$ and $0.35$ for $\alpha = 0^\circ, 5^\circ, 10^\circ$ and $15^\circ$, respectively. This value for the cone with $\alpha = 35^\circ$ was about $z/D = 0.3$. Further, the attenuation of down-flow velocity may affect the bed shear stress around the piers. Also, with the increase of the side slopes, the up-warding flow can be observed at the upside of the piers. As the down-flow is reduced in front of the pier, the bed shear stress is reduced remarkably.

Figure 7 shows the normalized time-averaged bed shear stress around the semi-conical piers with different $\alpha$. The values of the bed shear stresses have been normalized by reference bed shear stress at position $D = 5D_2$ at the upstream at the plane of symmetry of the domain. The maximum bed shear stress in the horizontal plane is located at the sides of the piers, as can be observed in this figure. The Large normalized bed shear stresses are seen for the cylinder, and the smallest values are observed for the semi-conical pier with $\alpha = 15^\circ$. Then the values of the shear stresses are decreased by increasing the $\alpha$. The reduction of the normalized bed shear stresses at the sides of the piers is evident. The maximum values of the normalized bed shear stress at the sides of the piers are $17.3, 12.57, 8.83,$ and $5.8$ for the piers with $\alpha = 0^\circ, 5^\circ, 10^\circ,$ and $15^\circ$, respectively. The reduction of the down-flow velocity and bed shear stress may regard as an important parameter to reduce the sediment transport discharge, and local scour at bridge piers.

The Time-averaged normalized bed shear stress at the symmetry line of the domain at the upstream and downstream of the piers have been compared in Fig. 8. The maximum normalized bed shear stress at this plane occurs in the vicinity of the up-side of the piers. A greater reduction of the normalized bed shear stress is observed with the increase of $\alpha$. Reducing the bed shear stress for the pier with $\alpha = 15^\circ$ is about $46\%$ compared to the cylinder. This reduction for the downstream of the pier is about $20\%$.

Figure 9 shows the mean (time-averaged) pressure coefficient ($C_p = (< p > - p_0)/0.5 \rho u_0^2$) calculated at the upstream of the piers at the symmetry line of the computational domain where $p_0$ is the static reference pressure at the inlet of the channel.

Figure 9a depicts the $C_p$ along the symmetry line of the flow near the bed. A line sharpness of $C_p$ is observed for all of the piers at $x/D_2 \approx 1.5$ for the cylinder. However, this sharpness gets closer to the pier, with the increase of $\alpha$. At this location, the main primary HVs are observed. Also, the curve becomes more horizontal with getting distance from the pier at $x/D_2 \approx -4.5$. 

---

**Fig. 6** Normalized time-averaged vertical velocity ($<w>$) at the upstream side of semi-conical piers at plane of symmetry $y/D_2 = 0$.
Figure 9b shows the pressure coefficient across the depth at just close to the upside of the piers. The overall trend of $C_p$ across the depth in Fig. 9b is similar to the experiment of [11] or numerical modeling of flow around pier by [48](not shown in this figure). The value of $C_p$ in the present study is slightly higher than the results obtained from the experiment of [11]. This discrepancy is due to differences between the pier model properties used in the experiment and the present study, particularly the values of $h/D$ and $R_D$. The minimum value of $C_p$ for the cylinder is 0.85. $C_p$ is decreased with the increase of $\alpha$ as shown in Fig. 9b.

Fig. 7 Time-averaged contours of the normalized bed shear stress around the piers (flow is from left to right)
3.3 Semi-conical piers, instantaneous values

3.3.1 Near wake

The cellular or multi-cell behavior cannot be observed in the present study due to the bed interaction and the small height of the piers. Streamlines of instantaneous flow have been shown in Fig. 10 for the downstream of the piers. The overall flow structure for the cylinder is vertical but, for semi-conical piers with the increase of $\alpha$, this shedding angle is increased. The method of calculating the oblique vortex shedding angle is carried out visually and is based on the vortices core angle behind the pier in most of the flow snapshots. Indeed, the value of this angle was averaged across multiple flow snapshots. This is a qualitative value, and no mathematical method was used for this issue. The angles of oblique vortex-shedding for the semi-conical piers is similar to those observed in the time-averaged three-dimensional streamlines behind the piers (not shown for brevity). The angle of inclination concerning the vertical axis for the semi-conical pier with $\alpha = 5^\circ$, $10^\circ$ and $15^\circ$ are about $25^\circ$, $37^\circ$ and $45^\circ$, respectively.

The full three-dimensionality of the flow, irregular and intermittent vortex shedding at the downstream of the piers cause highly fluctuating lift ($C_L = F_L / 0.5 \rho_u^2 A$) and drag ($C_D = F_D / 0.5 \rho_u^2 A$) forces on the cylinder surface (see Fig. 11). Where these coefficients are computed using the pressure and shear stresses acting on the pier surface. The drag coefficient for all the piers is close to 1. This value agrees with other researches about flow...
around cylinders [23, 30]. The lift and drag coefficients show varying amplitude and intensity, hence require substantially long averaging times. Fast Fourier transformation (FFT) was applied to the integrated lift forces.

The power spectrum density of this analysis based on the covariance spectral model is shown in Fig. 12. In this figure, the frequency of vortex shedding is decreased with the increase of $\alpha$. The frequency of the vortex shedding $f$ for semi-conical piers is 1.01, 1.0, 0.92, and 0.83 (Hz) for the piers with $\alpha = 0^\circ, 5^\circ, 10^\circ$ and $15^\circ$, respectively. The most reduction of $f$ is about 18% for the semi-conical pier with $\alpha = 15^\circ$ compared to the cylinder. The frequency of lift force in the present study for semi-conical piers is larger than the conical island in [41] due to the larger base diameter and larger $\alpha$ of the cone. The rate of the scouring process can be affected by the reduction of the $f$ as mentioned by researchers [15, 17]. Strouhal number for these piers ($St_f = fD/u_0$) are 0.21, 0.2, 0.19 and

---

![Fig. 10] Three-dimensional instantaneous stream-lines of the flow around the semi-conical pier with different side slopes

![Fig. 11] Lift and Drag coefficient of the flow around semi-conical piers. The upper curves correspond to the drag coefficient and the lower correspond to the lift coefficient
0.17 for the cylinder to semi-conical pier with \( \alpha = 15^\circ \), respectively. The value of \( \text{St}_D \) for the cylinder is comparable with its found in experiments [1, 37].

Figure 13 depicts the power spectrum density (PSD) of the lift coefficient as a function of frequency. This figure is produced using a snapshot of the flow field, gathered during eighteen vortex-shedding cycles. The spectrum exhibits a slope close to predicted by Kolmogorov’s theory. This agreement may show that the present LES model has predicted most of the energy in the turbulent scales of the flow.

### 3.3.2 Horseshoe vortices

As mentioned by researchers, the HV has an important role in the scour process [6, 11, 48]. The structure of the HV is divided into two laminar and turbulent regimes. Different regimes show different behaviors of the HV. These regimes depend on the Reynolds
number of the pier, $R_D$. HV from $R_D = 1000$ to $R_D = 14000$ are classified as laminar HV [29]. Some of the researchers [20, 29, 50] mentioned five sub-regime for the laminar HV system as Steady, Oscillatory, Amalgamating, Breakaway, and Transitional sub-regimes. Using the PIV method, [36] investigated the dynamical behavior of HVs in an experimental study of flow around an emerging obstacle. They suggested new categorized regimes for laminar HVs. In their experiments, the typology of HV was based on the dynamics of HVs and can be summarized into nine regimes. Also, two more regimes (semi-irregular and irregular regimes) exist for higher Reynolds numbers of the obstacle. They declared that their typology is similar to the one presented by [20]. The transitional vortex system in [20] is equivalent to an irregular regime. For very high Reynolds numbers, the bimodal behavior of the HV can be observed. $R_D$ for the present study is about 7600, which falls into the transitional sub-regime or irregular regime. HV system may be observed from the symmetry plane to the perpendicular plane to the flow. The process of changes for the HVs for the primary HV near the pier is intermittent. For the primary horseshoe vortex (PHV), the alternation is ended with a combination of the PHV and the secondary horseshoe vortex (SHV).

The successive instantaneous streamlines in front of the semi-conical piers in the vertical symmetry plane are depicted in Fig. 14. This figure depicts six snapshots of the time evolution of HVs over a full HV period. The arrows indicate the displacement of the vortex centre in HV from the previous location. The length of the HV region for the cylinder is longer than the others, as shown in this figure. The flow separation

\[ \text{Fig. 14 HV temporal variations upstream of semi-conical piers with different side slopes in the plane of symmetry. Streamlines produced from instantaneous streamwise and vertical velocities. a Cylinder, b Semi-conical pier } \alpha = 5^\circ, \text{ c Semi-conical pier } \alpha = 10^\circ, \text{ and d Semi-conical pier } \alpha = 15^\circ. \text{ The red arrows in the figure indicate the change in position of the vortices from the two consecutive snapshots.} \]
location for piers with \( \alpha = 0^\circ, 5^\circ, 10^\circ, \) and \( 15^\circ \) is \( x/D_2 = -1.9, -1.6, -1.4, \) and \( -1.2, \) respectively. The merging and diffusing of vortices have been observed on all of the piers. In an HVs period, the vortex merging occurs twice for the piers. The HV's overall shape changes over time and cannot be considered a sustainable form. There is no discernible difference in the number of vortices between the cylinder and semi-conical piers. However, the sizes of HVs are different. The height of the HVs increases with the increase of \( \alpha, \) but their central locations become closer to the piers. Increasing the height of the vortex corresponds to the results of [36], where increasing \( D_2/h \) increases the height of the vortices.

According to Launay et al. (2017), the HV regime in the current study for the cylinder is in a merging-diffusing regime with a breaking phase, and the breaking phase cannot be observed for piers with larger \( \alpha. \) In addition, as \( \alpha \) increases, the HV regime shifts from coherent to semi-irregular. This transition is characterized by the appearance of small-scale non-coherent structures (small-scale perturbations) in the HV. For pier with \( \alpha = 5^\circ, \) one diffusing phase at \( t = T/6, \) two successive merging in \( t = 2T/6, \) and \( t = 3T/6 \) are seen. For \( \alpha = 10^\circ \) diffusing phase is observed from the initial stage to \( t = 2T/6 \) and the merging phase is seen from \( t = 3T/6 \) to \( t = 5T/6. \) In addition, for \( \alpha = 15^\circ \) two merging phases are observed in \( t = 2T/6 \) and \( t = 4T/6. \) Overall, it appears that the behaviour of HV may not be classified as cylinders due to differences in shape and \( D/h, \) as these values in [36] \( (D/h = 1.0 \text{ to } 2.33) \) are significantly higher than present study \( (D/h = 0.4). \)

In the present study, at three points located upstream of the piers (in the x-z vertical plane), the instantaneous velocities have been recorded every 0.01 s. Figure 14a shows the location of these points. They are located in the area where the HVs have the highest spatial variations. The points are located at a distance of about 0.02 h to 0.06 h from the bed. The primary vortices are denoted by PV1 and PV2, and the vortex between the primary vortices is denoted by the bed vortices BV1 and BV2. The time-lapse of velocities was performed at time intervals of 0.02 s over a period of 30 s.

The power spectral density analysis over the time variation of the flow velocities for \( \alpha = 0^\circ, 10^\circ \) in points 1, 2, and 3 are presented in Fig. 15. To assess the effect of side slope on HVs frequencies, only a semi-conical pier with \( \alpha = 10^\circ \) was used. As can be observed from Fig. 15b, for the cylinder, the most energetic peak occurs for point 3. Most robust peaks correspond to points 2 and 1, respectively. The dominant frequency for point 3 is equal to 0.51 (Hz). The multiple peaks can be observed for point 1. It may show that at this point, temporal changes of the velocity are high. Due to the spatial variations of PV2 and BV1 and the placing of these vortices near point 1, several dominant frequencies are observed in the figure. The frequencies of the HVs for point 3 vary from 0.25 to 2 (Hz), which is slightly larger than the results obtained in the experiment of [11].

In Fig. 15c for the cone with \( \alpha = 10^\circ, \) the most dominant frequency of the HV for all of the points is equal to 0.6 (Hz). More recently in an experimental study [45], the spatiotemporal evolution of the turbulent HV was investigated. They declared that there was no meaningful relationship between Reynolds of the pier and HV dynamics. Based on the results of this experiment, the \( St_{D_2} = fD_2/u_0 \) varies from 0.06 to 0.25 for turbulent HV. In the current study, the frequency of HV for a semi-conical pier with \( \alpha = 10^\circ \) is equivalent to 0.21 (Hz), which agrees with the results of [30], and [45]. According to this figure, the most energetic oscillation corresponds to point 2. This point is located in the area in which the most robust horseshoe vortex (PV1) is located. Also, for points 2 and 3, some energetic frequencies are observed due to interactions of the HVs. Based on Fig. 15b, c, the strongest HV for the cylinder has more distance from the pier than the cone. For the cylinder pier, two or more dominant peaks are observed, and for the cone-shaped pier, one energetic peak.
frequency is observed for all of the points. The dominant frequency of the cone has more energy than that of the cylinder. It may be due to the pier shape and easier vortex formation in front of the cone than the cylinder.
3.4 Conclusion

This paper presented the results of a large-eddy simulation of turbulent flow around semi-conical piers. Downstream and upstream flow characteristics were the most important modeling results. The down-flow velocity was reduced for tapered piers by increasing the side slopes (α). The reduction was partially caused by an increase in α and part by a decrease in pressure gradient at the pier upstream. The numerical results showed a significant reduction in bed shear stress around the semi-conical piers, especially at upstream and downstream. These reductions for the semi-conical pier with α = 15° were about 46% and 20% at the upstream and downstream, respectively, if compared to the cylinder.

Instantaneous flow around the semi-conical piers indicated the oblique vortex shedding behind the piers. The inclination of the wakes was increased by increasing the side slope of the piers. The integrated lift forces on the semi-conical piers due to vortex shedding showed a dominant frequency. With the increase in α, the frequency of vortex shedding decreased.

The size and location of the HVs changed with the side slope of the piers. The primary vortex had the most significant and energetic frequencies. The value of frequency was equal to 0.51 and 0.6 (Hz) for the cylinder and semi-conical pier α = 10°, respectively. Overall, the results of this study indicated that the flow around the semi-conical pier was different from the cylinder. Some flow behaviors around the semi-conical piers may affect the scour process, as mentioned by other researchers in their experiments.

Author Contributions All authors contributed to the study conception and design. Material preparation, data collection and analysis were performed by Yasin Aghaee-Shalmani and Habib Hakimzadeh. The first draft of the manuscript was written by Yasin Aghaee-Shalmani and all authors commented on previous versions of the manuscript. All authors read and approved the final manuscript. Conceptualization: Yasin Aghaee-Shalmani; Methodology: Yasin Aghaee-Shalmani; Formal analysis and investigation: Yasin Aghaee-Shalmani; Writing—original draft preparation: Yasin Aghaee-Shalmani; Writing—review and editing: Yasin Aghaee-Shalmani and Habib Hakimzadeh; Resources: Yasin Aghaee-Shalmani; Software: Yasin Aghaee-Shalmani; Supervision: Habib Hakimzadeh.

Funding The authors did not receive support from any organization for the submitted work. No funding was received to assist with the preparation of this manuscript. No funding was received for conducting this study. No funds, grants, or other support was received.

Declarations

Conflicts of interest The authors have no relevant financial or non-financial interests to disclose. The authors have no conflicts of interest to declare that are relevant to the content of this article. All authors certify that they have no affiliations with or involvement in any organization or entity with any financial interest or non-financial interest in the subject matter or materials discussed in this manuscript. The authors have no financial or proprietary interests in any material discussed in this article.

References

1. Achenbach E, Heinecke E (1981) On vortex shedding from smooth and rough cylinders in the range of reynolds numbers 6×10^4 to 5×10^6. J Fluid Mech 109:239–251
2. Afzal MS, Bihs H, Kumar L (2020) Computational fluid dynamics modeling of abutment scour under steady current using the level set method. Int J Sedim Res 35(4):355–364
3. Aghaee Y, Hakimzadeh H (2010) Three dimensional numerical modeling of flow around bridge piers using les and rans. River Flow 2010:211–218
4. Aghaee-Shalmani Y, Hakimzadeh H (2015) Experimental investigation of scour around semi-conical piers under steady current action. Eur J Environ Civ Eng 19(6):717–732
5. Aghaee-Shalmani Y, Hakimzadeh H (2016) Numerical modeling of 2-d and 3-d flows using artificial compressibility method and collocated mesh. J Appl Fluid Mech 9(5):2333–2345
6. Baker C (1980) The turbulent horseshoe vortex. J Wind Eng Ind Aerodyn 6(1–2):9–23
7. Bozkus Z, Yildiz O (2004) Effects of inclination of bridge piers on scouring depth. J Hydraul Eng 130(8):827–832
8. Breuer M, Hänel D (1993) A dual time-stepping method for 3-d, viscous, incompressible vortex flows. Comput Fluids 22(4–5):467–484
9. Cheng Y, Lien F, Yee E, Sinclair R (2003) A comparison of large eddy simulations with a standard k-ε Reynolds-averaged navier-stokes model for the prediction of a fully developed turbulent flow over a matrix of cubes. J Wind Eng Ind Aerodyn 91(11):1301–1328
10. Cubero A, Fueyo N (2007) A compact momentum interpolation procedure for unsteady flows and relaxation. Numer Heat Transf Part B Fundam 52(6):507–529
11. Dargahi B (1989) The turbulent flow field around a circular cylinder. Exp Fluids 8(1):1–12
12. Dargahi B (1990) Controlling mechanism of local scouring. J Hydraul Eng 116(10):1197–1214
13. Devenport WJ, Simpson RL (1990) Time-depeendent and time-averaged turbulence structure near the nose of a wing-body junction. J Fluid Mech 210:23–55
14. Dey S, Raikar RV (2007) Characteristics of horseshoe vortex in developing scour holes at piers. J Hydraul Eng 133(4):399–413
15. Ettema R, Melville BW, Barkdoll B (1998) Scale effect in pier-scour experiments. J Hydraul Eng 124(6):639–642
16. Fredsøe J, Sumer BM (1997) Scour at the round head of a rubble-mound breakwater. Coast Eng 29(3–4):231–262
17. Fredsøe J, Sumer BM (2002) The mechanics of scour in the marine environment, vol 17. World Scientific Publishing Company, Singapore
18. Fu WS, Li CG, Lin WF, Chen YH (2009) Roe scheme with preconditioning method for large eddy simulation of compressible turbulent channel flow. Int J Numer Meth Fluids 61(8):888–910
19. Gazi AH, Afzal MS (2020) A review on hydrodynamics of horseshoe vortex at a vertical cylinder mounted on a flat bed and its implication to scour at a cylinder. Acta Geophys 68:861–875
20. Greco JJ (1990) The flow structure in the vicinity of a cylinder-flat plate junction: flow regimes, periodicity, and vortex interactions. Ph.D. thesis, Lehigh University
21. Guan D, Chiew YM, Wei M, Hsieh SC (2019) Characterization of horseshoe vortex in a developing scour hole at a cylindrical bridge pier. Int J Sedim Res 34(2):118–124
22. Gushchin V, Kostomarov A, Matyushin P, Pavlyukova E (2002) Direct numerical simulation of the transitional separated fluid flows around a sphere and a circular cylinder. J Wind Eng Ind Aerodyn 90(4–5):341–358
23. Hansen R, Forsythe J (2003) Large and detached eddy simulations of a circular cylinder using unstructured grids. In: 41st Aerospace Sciences Meeting and Exhibit, p 775
24. Hsiao F, Pan J, Chiang C (1993) The study of vortex shedding frequencies behind tapered circular cylinders. ASME-PUBLICATIONS-PVP 245:103–103
25. Hsiao FB, Chiang CH (1998) Experimental study of cellular shedding vortices behind a tapered circular cylinder. Exp Thermal Fluid Sci 17(3):179–188
26. Kahraman A, Sahin B, Rockwell D (2002) Control of vortex formation from a vertical cylinder in shallow water: effect of localized roughness elements. Exp Fluids 33(1):54–65
27. Khosla P, Rubin S (1974) A diagonally dominant second-order accurate implicit scheme. Comput Fluids 2(2):207–209
28. Kim WW, Menon S (1999) An unsteady incompressible Navier–Stokes solver for large eddy simulation of turbulent flows. Int J Numer Methods Fluids 31(6):983–1017
29. Kirkil G, Constantinescu G (2012) A numerical study of the laminar necklace vortex system and its effect on the wake for a circular cylinder. Phys Fluids 24(7):073602
30. Kirkil G, Constantinescu G (2015) Effects of cylinder Reynolds number on the turbulent horseshoe vortex system and near wake of a surface-mounted circular cylinder. Phys Fluids 27(7):075102
31. Kirkil G, Constantinescu S, Ettema R (2008) Coherent structures in the flow field around a circular cylinder with scour hole. J Hydraul Eng 134(5):572–587
32. Kirkil G, Constantinescu G, Ettema R (2009) Detached eddy simulation investigation of turbulence at a circular pier with scour hole. J Hydraul Eng 135(11):888–901
33. Kitagawa T, Ohta H (2008) Numerical investigation on flow around circular cylinders in tandem arrangement at a subcritical Reynolds number. J Fluids Struct 24(5):680–699
34. Labbé D, Wilson P (2007) A numerical investigation of the effects of the spanwise length on the 3-d wake of a circular cylinder. J Fluids Struct 23(8):1168–1188
35. Lam K, Lin Y (2008) Large eddy simulation of flow around wavy cylinders at a subcritical Reynolds number. Int J Heat Fluid Flow 29(4):1071–1088
36. Launay G, Mignot E, Riviere N, Perkins R (2017) An experimental investigation of the laminar horseshoe vortex around an emerging obstacle. J Fluid Mech 830:257–299
37. Lienhard JH et al (1966) Synopsis of lift, drag, and vortex frequency data for rigid circular cylinders, vol 300. Technical Extension Service, Washington State University Pullman, WA
38. Nagata N, Hosoda T, Nakato T, Muramoto Y (2005) Three-dimensional numerical model for flow and bed deformation around river hydraulic structures. J Hydraul Eng 131(12):1074–1087
39. Narasimhamurthy VD, Andersson H, Petersen B (2009) Cellular vortex shedding behind a tapered circular cylinder. Phys Fluids 21(4):044106
40. Olsen NR, Melaæen MC (1993) Three-dimensional calculation of scour around cylinders. J Hydraul Eng 119(9):1048–1054
41. Ouro P, Wilson CA, Evans P, Angeloudis A (2017) Large-eddy simulation of shallow turbulent wakes behind a conical island. Phys Fluids 29(12):126601
42. Paik J, Escauriaza C, Sotiropoulos F (2007) On the bimodal dynamics of the turbulent horseshoe vortex system in a wing-body junction. Phys Fluids 19(4):045107
43. Parnaudeau P, Heitz D, Lamballais E, Silvestrini JH (2005) Direct numerical simulations of vortex shedding behind cylinders with spanwise linear nonuniformity. In: Fourth international symposium on turbulence and shear flow phenomena, Begel House Inc
44. Pourahmadi M, Hakimzadeh H (2011) Experimental investigation of local scour around conical pier. In: International conference on offshore mechanics and arctic engineering, vol 44380, pp 351–355
45. Qi WG, Liu J, Gao FP, Li B, Chen QG (2022) Quantifying the spatiotemporal evolution of the turbulent horseshoe vortex in front of a vertical cylinder. Phys Fluids 34(1):015110
46. Rodi W (1997) Comparison of les and rans calculations of the flow around bluff bodies. J Wind Eng Ind Aerodyn 69:55–75
47. Rodi W (2006) Dns and les of some engineering flows. Fluid Dyn Res 38(2–3):145
48. Rouland A, Sumer BM, Fredsøe J, Michelsen J (2005) Numerical and experimental investigation of flow and scour around a circular pile. J Fluid Mech 534:351
49. Salaheldin TM, Imran J, Chaudhry MH (2004) Numerical modeling of three-dimensional flow field around circular piers. J Hydraul Eng 130(2):91–100
50. Seal C, Smith C, Rockwell D (1997) Dynamics of the vorticity distribution in endwall junctions. AIAA J 35(6):1041–1047
51. Sheu TW, Lin RK (2004) Newton linearization of the incompressible Navier–Stokes equations. Int J Numer Methods Fluids 44(3):297–312
52. Sumer B, Fredsøe J, Christiansen N, Hansen S (1995) Bed shear stress and scour around coastal structures. Coast Eng 1994:1595–1609
53. Unger J, Hager WH (2007) Down-flow and horseshoe vortex characteristics of sediment embedded bridge piers. Exp Fluids 42(1):1–19
54. Vaghefi M, Ghodsian M, Salimi S (2016) The effect of circular bridge piers with different inclination angles toward downstream on scour. Sadhana 41(1):75–86
55. Valles B, Andersson H, Jenssen C (2002) Oblique vortex shedding behind tapered cylinders. J Fluids Struct 16(4):453–463
56. Williamson CH (1996) Vortex dynamics in the cylinder wake. Annu Rev Fluid Mech 28(1):477–539
57. Zhang W, Zapata MU, Bai X, Pham-Van-Bang D, Nguyen KD (2020) Three-dimensional simulation of horseshoe vortex and local scour around a vertical cylinder using an unstructured finite-volume technique. Int J Sedim Res 35(3):295–306

Publisher’s Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.