Unstable flow structure around partially buried objects on a simulated river bed

Yovanni A. Cataño-Lopera, Blake J. Landy and Marcelo H. García

ABSTRACT

The unsteady flow characteristics around two partially buried objects, a short cylinder and a truncated cone, were examined with a three-dimensional, non-hydrostatic hydrodynamic model under similar steady unidirectional currents with flow Reynolds numbers, $R_e$, of 86,061 and 76,209, respectively. Model simulations were conducted with the two objects partially buried in a simulated rippled river bed. A Reynolds-averaged Navier–Stokes (RANS) equation model coupled with a $\kappa$-$\epsilon$ turbulence closure was used to validate the experimental velocity measurements. A large eddy simulation (LES) turbulence model was subsequently used to characterize the unsteady flow structure around the objects. The LES closure allowed for the characterization of highly unsteady coherent turbulent structures such as the horse-shoe vortex, the arch-shaped vortex, as well as vortex shedding in the wake of the object.

Key words | LES RANS numerical modeling, object burial, scour hole, sediment transport, turbulent flow

NOTATION

- $A_i$: fractional area open to flow in the $i$-direction
- $a$: 0.247 is a constant
- $C_p$: 0.09 is a constant
- $C_s$: 0.2 is the Smagorinsky constant
- $d_5$: median diameter of sand
- $D$: diameter of the cylinder
- $D_{bc}$: base diameter of truncated cone
- $D_{tc}$: top diameter of truncated cone
- $F$: fluid fraction function
- $f_i$: viscous accelerations
- $G_i$: body accelerations
- $g$: acceleration due to gravity
- $h$: water depth
- $h_c$: height of truncated cone
- $L_c$: length of the short cylinder
- $P$: pressure
- $R_e$: Reynolds number
- $R_{ed}$: object Reynolds number
- $R_l$: length of the recirculation zone
- $S_i$: length of the separation zone
- $S_t$: Strouhal number
- $T_s$: period of the vortex shedding oscillation
- $u_i$: velocity
- $U_{nd}$: dimensionless streamwise velocity
- $V_f$: fractional volume open to flow
- $u^*$: shear velocity
- $y_0$: distance from the solid wall to the location of tangential velocity
- $x$, $y$, $z$: spatial coordinates
- $\rho$: water density
- $\nu$: molecular viscosity
- $\nu_T$: turbulent viscosity
- $\kappa$: turbulent kinetic energy
- $\kappa_v$: 0.41 is the Von Kármán constant
- $\kappa_s$: equivalent wall roughness height
- $\gamma_c$: specific gravity
- $\epsilon$: dissipation rate of turbulent energy
- $\epsilon_{ij}$: strain rate tensor

doi: 10.2166/hydro.2016.060
\[ \Delta \] turbulent characteristic length scale  
\[ \delta x, \delta y, \text{and} \delta z \] cell size in the three Cartesian directions  
\[ \tau_{bp} \] bed shear stress  
\[ \tau_{ij} \] shear stress tensor

**INTRODUCTION**

The use of numerical simulations to investigate the scour mechanisms around two-dimensional (2D) and three-dimensional (3D) objects on river-like beds has increased substantially over the last decade (Liu & García 2007; Liu et al. 2008). Early efforts studied the hydrodynamics of the flow around objects mounted on a flat immobile bed. Lately, the efforts have been extended to the study of the flow structure over scoured regions around a variety of objects. Over time, the numerical models have become more sophisticated to deal with more complex flow scenarios (e.g., fluid–sediment–object interaction), while the availability of experimental data increases. Numerical simulations of local scour and lee-wake scouring of offshore pipelines and other objects have been extensively analyzed using Reynolds-averaged Navier–Stokes (RANS) equations with both \( \kappa-\epsilon \) and \( \kappa-\omega \) turbulence closure models. Salaheldin et al. (2004) used FLUENT to simulate the flow field around circular piers on a scoured bed using a RANS model with a \( \kappa-\epsilon \) turbulence closure model. Rouland et al. (2005) used a \( \kappa-\omega \) turbulence closure to characterize the steadiness and unsteadiness of the flow field.

Thanks to the rapid increase in computer power and equation solver schemes, other numerical techniques have been explored in recent years, allowing more detailed investigation of the unsteadiness associated with the coherent flow structures around diverse obstacles. Chrisohoides et al. (2003) studied the vortices induced by a vertical bridge abutment upon a flat bed using unsteady RANS along with a \( \kappa-\omega \) turbulence closure scheme. Paik et al. (2007) used detached eddy simulation (DES) to analyze the bimodal dynamics of the horseshoe vortex (HV) at the base of a wing–body junction. Although DES reproduced fairly accurately the main characteristics of the experimental flow field reported by Devenport & Simpson (1990), the model was unable to accurately reproduce the location of the HV structure. Addition of artificial turbulent fluctuations at the inflow of the numerical domain was needed to improve the simulated HV location.

Large eddy simulation (LES) models have also been applied in the case of flow around piers as presented by Tseng et al. (2000), Choi & Yang (2002), and Kirkil et al. (2008). Kirkil et al. (2008) investigated the flow field and turbulent coherent structures around an experimental circular cylinder with a developed scour hole via LES simulations. Kirkil et al. (2008) analyzed extensively the HV upstream of the object and the vortex shedding in the wake of the object. These authors identified regions of coexisting counter-rotating zones upstream of the object accompanied by complex flow unsteadiness and shear stress distributions responsible for near-object scouring. Huang et al. (2009) conducted LES simulations of the scale effects on turbulent flows and scour around large bridge piers using FLUENT; they concluded that the simulations reproduced qualitatively well the main features of the flow pattern around the object.

Other obstacle geometries that have been examined with LES, employing a range of sub-grid-scale and boundary formulations, include cubes (Rodi 1997; Krajnović & Davidson 2002), hemispheres (Manhart 1998), and spheres (Truelsen et al. 2005). Generally, when comparing the predictive capabilities of RANS-based models versus LES simulations, it has been shown that both approaches reproduce fairly well the length of the vortex shedding region in the wake of the object when compared to experimental data (Smith & Foster 2007). However, the variation in the shedding amplitude was not well captured when using the standard \( \kappa-\epsilon \) closure scheme. The simulations conducted by Rodi (1997) of flow around a surface-mounted cube using any of the \( \kappa-\epsilon \) formulations under-predicted upstream separation and downstream reattachment lengths. LES-based model predictions were in better agreement with laboratory data; however, there was some variability in the predictions depending on the assumed boundary layer formulation. Rodi (1997) concluded that LES is better suited to reproduce the coherent structures, such as the HV system, in flow fields around piers upon flat or scoured beds, compared to other computational techniques such as RANS. In LES, the unsteady dynamics of the energetically important scales in the flow is directly computed and only
the effect of the small (filtered) scales on the resolved scales is modeled (Kirkil et al. 2008).

The main goal of the present study was to improve our understanding of the hydrodynamics around partially buried objects and to characterize the mechanism of HV and vortex shedding in the wake, which is responsible for erosion, transport, and deposition of sediment in the vicinity of the object. In the present study, modeled flow is contrasted against experimental mean velocities and turbulent flow structures around partially buried objects under unidirectional steady flow, described in Cataño-Lopera et al. (2015). The results show that beyond the mean flow characteristics, flow unsteadiness plays a major role in determining sediment mobility and the extent and development of the scour hole. The present findings are intended to assist the development of mathematical models of the local scour and burial mechanisms of 3D objects, in particular of short cylinders (SC) and truncated cones (TC). The applications of the current findings are expected to be helpful for future military and civilian welfare.

METHODS

The numerical model

The three-dimensional, non-hydrostatic hydrodynamic model FLOW-3D was used to simulate the flow field around two 3D objects. This model allows for solving the motion of fluids to obtain transient, three-dimensional solutions to multi-scale, multi-physics flow problems (Flow Science Inc. 2009). The model resolves fluid–fluid, and fluid–air interfaces with a non-boundary fitted rectangular grid and a volume of fluid (VOF) approach that resolves the grid cells into separate fractional fluid components containing the fraction of fluid and non-fluid fraction in the cell (Hirt & Nichols 1981). The fluid fraction function \( F \) is defined to be equal to 1.0 in the fluid and 0.0 outside the fluid, i.e., in the void (Flow Science Inc. 2009). Similarly, a fractional area–volume obstacle representation (FAVOR) approach is used to parameterize the flow within cells that contain fluid–obstacle boundaries. FAVOR uses three-dimensional quadratic equations to define complex obstacle geometries specified separately from the mesh. The model solves the 3D Navier–Stokes momentum equation for incompressible flow simultaneously with the mass continuity equation. The momentum equation, without considering terms describing flow losses in porous media or across baffle plates and terms related to mass sources or sinks, can be written as:

\[
\frac{\partial u_i}{\partial t} + \frac{1}{V_f} \left( A_i u_i \frac{\partial F}{\partial x_i} \right) = - \frac{1}{\rho} \frac{\partial P}{\partial x_i} + G_i + f_i
\]

where, \( \rho \) is the water density, \( A_i \) is the fractional area of the object. In the present study, modeled flow is contrasted against experimental mean velocities and turbulent flow structures around partially buried objects under unidirectional steady flow, described in Cataño-Lopera et al. (2015). The results show that beyond the mean flow characteristics, flow unsteadiness plays a major role in determining sediment mobility and the extent and development of the scour hole. The present findings are intended to assist the development of mathematical models of the local scour and burial mechanisms of 3D objects, in particular of short cylinders (SC) and truncated cones (TC). The applications of the current findings are expected to be helpful for future military and civilian welfare.

In Equations (1) and (2), \( u_i \) is the velocity, \( P \) is the pressure, \( \rho \) is the water density, \( A_i \) is the fractional area open to flow in the \( i \)-direction (index is not used to imply summation), \( V_f \) is the fractional volume open to flow, \( G_i \) represents the body accelerations, such as gravity, \( f_i \) represents the viscous accelerations, \( \tau_{ij} \) is the shear stress tensor, and \( \tau_{b,i} \) is the wall shear stress that is represented by:

\[
\tau_{ij} = -\rho (v + v_T) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

where, \( \rho (v + v_T) \) is the total dynamic viscosity, assuming that it is given by the summation of the molecular and turbulent viscosities, \( v \) and \( v_T \), respectively.

Since the present study explores two turbulent closure schemes, \( \kappa - \varepsilon \) and LES, a short summary of the eddy viscosity closures used in both schemes is presented.

The widely used two transport equation model, \( \kappa - \varepsilon \), considers transport equations for the turbulent kinetic energy \( \kappa = 1/2 (u'^2 + v'^2 + w'^2) \) in which the apostrophes (‘) indicate turbulent velocity fluctuations; and the dissipation rate of turbulent kinetic energy \( \varepsilon \) (Harlow & Nakayama 1967). The equations of motion are closed with the standard closure for the eddy viscosity given by:

\[
\rho v_T = \frac{\rho \kappa^2}{\varepsilon}
\]
The closure equations for $\kappa$ and $\varepsilon$ with standard coefficients are given in Wilcox (2000). The boundary conditions for the $\kappa$ and $\varepsilon$ equations are defined with a logarithmic law of the wall formulation given by:

$$
\kappa = \frac{u^2}{\sqrt{C_\mu}}, \quad \varepsilon = \frac{u^5}{\kappa y_0}
$$  \tag{5}

where, $C_\mu = 0.09$ and $y_0$ is the distance from the solid wall to the location of tangential velocity, $u$. Notice here that the dynamic viscosity approximation in Equation (3) is good for flows with high levels of turbulence, i.e., when the turbulent viscosity is much larger than the molecular viscosity (Harlow & Nakayama 1967).

After dropping tensor notation, the shear velocity, $u_s$, is solved iteratively at no-slip boundaries with the combined smooth and rough logarithmic law of the wall equation given by:

$$
u = u_s \left[ 1 \left( \frac{\rho u_s y_0}{\mu + \rho u_s \kappa_s} \right) + 5.0 \right]
$$  \tag{6}

where, $\kappa_s = 0.41$ is the Von Kármán constant, $a = 0.247$ is a constant, and $\kappa_s$ is the equivalent wall roughness height. Since the FAVOR method does not precisely locate walls within a cell, approximations must be introduced to find $u_s$, $u$, and $y_0$ (Flow Science Inc. 2009). For this, the normal direction of the wall in the cell is determined, and then $u$ is computed as the component of the cell-centered velocity parallel to the wall. The average distance to the wall, $y_0$, is defined as half of the cell width in the wall normal direction. Finally, $u_s$ is determined iteratively from Equation (6) in terms of $u$ and $y_0$.

Notice that the denominator of the logarithmic term in Equation (6) represents an effective viscosity that characterizes the effect of the rough boundary. The wall shear stress is defined as:

$$
\tau_b = 2\rho |u_s| u_s
$$  \tag{7}

which is included directly into the momentum Equation (1) via Equation (3).

The second turbulence closure scheme used in this study is LES. This approach uses Kolmogorov’s theory of self-similarity, which considers that the large eddies in the flow are dependent on the flow domain geometry while the smaller scales are more universal. The closure scheme considered in this study uses the Smagorinsky approximation (Wilcox 2000) with an eddy viscosity formulation given by:

$$
\mu_T = \rho (C_s \Delta)^2 \sqrt{\varepsilon_0 e_{ii}}
$$  \tag{8}

where, the Smagorinsky constant, $C_s$, usually falls within the interval 0.1 to 0.2. In the present case, $C_s = 0.2$, and $e_{ii}$ is the strain rate tensor given by:

$$
\varepsilon_{ii} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
$$  \tag{9}

and the characteristic length scale is defined as:

$$
\Delta = (\delta x \delta y \delta z)^{1/3}
$$  \tag{10}

where, $\delta x$, $\delta y$, and $\delta z$ are the cell sizes in the three Cartesian directions. The wall shear stress, $\tau_b$, is defined differently than in the $\kappa$-$\varepsilon$ model as:

$$
\tau_b = \frac{\mu + \rho u_k \kappa_s u_s}{y_0}
$$

along each coordinated axis. As previously, the computed wall shear stress is directly included in the solution of the momentum Equation (1) through Equation (3). The wall shear approximation is only applied in cells where an obstacle blockage is present, i.e., where $V_f \neq 1$. In cells without an obstacle, the wall shear stress is zero.

**Experimental setup**

The experiments were conducted on a 49 m long, 1.83 m wide, and 1.2 m deep flume with a steel floor and acrylic sidewalls located at the Ven Te Chow Hydrosystems Laboratory, University of Illinois at Urbana-Champaign. The SC and the TC were initially placed upon a flat sand pit 0.5 m deep, 24 m long, and 1.83 m wide. The object was placed at the center of the flume at a distance of 13 m from the origin of the sand pit, adequate for flow.
development. The sediment was a well-sorted size distribution of silica sand with median diameter $d_{50} = 0.25$ mm. A water depth of $h = 0.41$ m above the initial flat bed was used in the experiments. The diameter, $D$, and length, $L_c$, of the SC are 10 cm and 20 cm, respectively. The base diameter, $D_{bc}$, top diameter, $D_{tc}$, and height, $h_c$, of the TC are 20 cm, 10 cm, and 20 cm, respectively. The specific gravity, $\gamma_c$, of the SC and TC were 2.3 and 2.0, respectively.

A 3D SonTek 10 MHz Acoustic Velocimetry (ADV) probe was used to measure the velocity field around the object after equilibrium morphodynamic conditions (EMC) were achieved. At the EMC stage, zones of sediment scouring and deposition around the object were developed along far-field sand ripples. Velocity measurements were recorded during 2 minutes in regions away from the object, and during 3 minutes in regions near the object characterized by relatively higher shear stresses and flow reversals. An extensive analysis of the velocity signals recorded with the ADV was conducted to ensure the measurements were of adequate quality to describe mean flow velocities and turbulent parameters (Cataño-Lopera et al. 2015).

**Numerical setup**

The numerical setup consists of the recorded bathymetry of the tandem sand bed plus object at equilibrium conditions embedded in a rectangular structured mesh, as shown in Figure 1. The mesh consisted of cells of variable size around the object, transitioning from finer cells in the object's vicinity to linearly increasing coarser cells towards the boundary of the domain.

Mesh sensitivity analyses to ensure mesh size independence simulation results were conducted for both RANS and LES cases. In the RANS case, initially two mesh sizes were tested. The first one consisted of 0.5 cm sided regular cubic cells spanning about five object diameters around the object in the streamwise $x$-direction, three diameters in the transverse $y$-direction, and two diameters in the vertical $z$-direction. A second mesh consisted of 0.3 cm sided regular cubic cells covering a similar spatial domain. In both cases, the cell sizes increased linearly towards the edges of the flow domain (Figure 1). For both objects, the coarser and finer grids used a total of 2,730,000 and 4,080,000 cells, respectively. The simulations showed that the mean flow structure, mean velocities, and turbulent kinetic energy...
were in better agreement with the experimental data for the finer mesh cases, but differences were negligible (Cataño-Lopera et al. 2015). The analysis presented hereafter corresponds to the case of simulations conducted with the finer grid.

In the LES case, in addition to the two meshes of 2,730,000 and 4,080,000 cells used in the RANS case, a third and finer mesh was evaluated. This mesh consisted of 0.2 cm sided regular cubic cells in the vicinity of the object and totaling about 4,925,000 cells. Characteristics of coherent turbulent structures, such as size and location of the HV, did not appreciably vary with respect to those associated with the intermediate mesh. Furthermore, the periodicity of the vortex shedding oscillations practically remained unaltered. LES simulation results presented in this study correspond to the finest mesh.

In both cases, a free surface boundary condition was imposed at the top of the domain for a mean water depth of 41 cm, measured from the initial bed level. A symmetry boundary condition was imposed at the lateral boundaries of the domain. The bottom of the domain was set to non-slip boundary conditions and consisted of the measured bathymetry, for which a roughness length of \( \kappa_s = 0.254 \text{ mm} \), equivalent to the mean grain size of the sand bed, was used. This roughness length was also set for the bathymetric region corresponding to the object. This was a reasonable assumption, since the two objects tested had a rough concrete surface with mean roughness length of 0.3 mm. At the downstream end of the computational domain a continuative outflow BC was specified.

A uniform flow field with mean velocity equal to the mean flow velocity was set as initial conditions (IC). In the RANS case, streamwise velocities and turbulent intensities obtained at the outflow boundary from a preliminary RANS simulation of a fully developed flow over a flat bed were set at the inflow boundary. In the LES case, turbulent velocity fluctuations at the outflow from a separate simulation over a similar channel with a flat bed were obtained. The LES velocity fluctuations were then fed through to the inflow boundary during the duration of the simulation. In the preliminary simulations, uniform mean velocity profiles were 21.80 cm/s and 19.38 cm/s as upstream BC for the SC and TC, respectively, while all other BCs were set as described above for the two partially buried objects.

Model simulations were run for about 70 s in the case of \( \kappa-e \), which allowed for proper flow development. In the case of the LES, longer simulation times of about 120 seconds were needed in order to properly characterize the unsteadiness and amplitude of the vortex shedding in the wake of the object. In each case, notice that the time necessary to allow for development of the flow field was about 6 seconds, which is basically the time it takes for a particle initially located at the upstream end of the computational domain to reach the outflow boundary.

The present simulations consider clear water conditions in which interaction of sediment particles with the flow coherent structures is neglected. This is supported by the findings from studies that have determined that the influence of sediment particles on the flow turbulence in the vicinity of the object and within the scour hole is negligible for low sediment concentrations (Kirke et al. 2008). Low sediment concentrations are distinctive of the conditions at EMC, during which the acoustic Doppler velocimeter (ADV) measurements were carried out. A detailed description of the experimental setup and test conditions is provided in Cataño-Lopera et al. (2015).

FLOW-3D uses a variable time step to preserve numerical stability at all times via the Courant number criterion. Simulations were run on a seven-node quad-core, dual-CPU Intel Xeon E5345 cluster. Each simulation was run on a node with 32 GB RAM. The total CPU time was about 600 hours of wall-clock time for the LES simulations for each object. The computational time for the \( \kappa-e \) simulations was considerably shorter.

The dynamics of the flow around the object

The computational fluid dynamics (CFD) simulations provide a more comprehensive picture of the flow structure around the object than that allowed by the measurements. The ADV measurements were conducted at a single point at a time that does not allow instantaneous plane view analysis of flow structures. On the other hand, CFD allows observation of time snapshots of the flow structure at any given plane (i.e., \( xy, xz, yz \)) and also its 3D time evolution, as will be shown later on (Figures 2–6). As
revealed by the experiments, the simulations show that the flow field is greatly disturbed in proximity to the object and extends for several object diameters downstream (DS). This is a consequence of the complex interaction between the approaching flow, already disturbed by the small-scale ripples, and the deformed flow around the object. Bedforms in the vicinity of the object make it even more difficult to predict the flow field when compared to flat bed conditions.

In general, scour initiates in the accelerated flow regions on the upstream (US) and lateral sides of the object and in the wake of the object. Shortly after, the bed scouring induced by the HV US system and at the lateral sides of the object becomes dominant. As in the case of scour around vertical
piers (Olsen & Kjellesvig 1998; Kirkil et al. 2008), while the scour hole increases, so does the size of the HV, but the bed shear stress decreases (Smith & Foster 2007). Once bed shear stresses reach values corresponding to local threshold values for sediment entrainment, scouring of the bed ceases and the flow reaches EMC. In reality, even at this condition some local erosion and deposition take place (Roulund et al. 2008), but the overall scour hole shape does not change.

Distinctive flow features around the object were observed. US of the object, sets of vortices develop near the bed as the flow separates. The interaction of these structures leads, in turn, to formation of larger HV patterns. In the case of the SC, strong flow separation occurs at the sides of the object, due partially to the sharp edges.

Similar to the case of a sphere placed on an erodible seabed (Truelsen et al. 2005), the object causes the flow to undergo substantial deformation due to the object blockage effect. Local acceleration of the flow occurs US of the object, enhancing the flow capacity to erode the bed, while the local Shields parameter increases. Sediment is

![Figure 3](https://iwaponline.com/jh/article-pdf/19/1/31/391074/jh0190031.pdf)
detached from the bottom and transported by the flow away from the object through deformed streamlines around the edges and above the object.

The HV is caused by the rotation of the incoming flow, while the adverse pressure gradient causes the approaching boundary layer to separate on the US side of the object. As the flow continues laterally around the edges, the limbs of the HV deform and extend DS. A second vortex is formed due to the flow separation from the top and sides of the obstacle. This arch-shaped vortex (AV) extends around the perimeter

---

**Figure 4** | Snapshots of the centerline (y = 0 cm) y-plane vorticity obtained with the LES simulations in the TC case at different times. The velocity vector arrows are superposed.
of the obstacle, with a rotation that entrains fluid into the recirculating region along the centerline of the object (i.e., \( y = 0 \) cm). Near the bed, this results in two counter-rotating vortices that may serve as a dominant DS depositional mechanism. As the AV grows in size, it is shed away from the obstacle by the exterior flow. This effectively removes the vortex from its generation mechanism, and a new vortex begins to form. Simultaneously, the lower edges of the AV start to interact with the tails of the HV, which allows the shed vortex to be ejected up into the water column. The downstream HV extensions contract as the streamlines contract in the lee of the obstacle (Smith & Foster 2007). Also, a recirculation region is formed in the wake of the object. Such recirculation regions display rather weak patterns at the final stages in which the Shields parameter is small, facilitating deposition of material transported from US and lateral sides of the body. In this way, the central sand ridge observed in Figure 2 is formed. The velocity vectors in this zone are mostly downward and curve towards the object. This helps explain the accumulation of sediment in this zone and the maximum height of the ridge being located close to the object. Notice also that the material accumulated in this region helps to counteract the backwards tilting tendency of the object as it loses support due to the scour hole developed underneath and US of it.

RESULTS AND DISCUSSION

Mean velocity flow field validation results

Figure 2 shows a comparison between modeled (left panels) and measured (right panels) dimensionless streamwise
velocities, $U_{nd}$, at the $x = 0$ cm cross section (top) and at $x = 10$ cm (bottom), for both the SC and TC, respectively. In general, the match is fairly good in terms of velocity magnitude, particularly far from the solid boundary. Near the object and bed boundaries, the simulated velocities along $x = 0$ cm for the SC case get very close to zero, as expected due to the wall presence. At $x = 10$ cm, in the TC case, the numerical simulations reveal regions of negligible velocities near the boundaries and velocities approaching the free stream velocity, $U_{nd} = 1.0$, in regions away from the object. The simulations successfully capture the recirculation zone, as indicated by negative velocities. The transverse extension of this region is still in very good agreement with the experiments. Nevertheless, the vertical extension appears to be larger in the simulations. A fully detailed comparison between experimental and simulated steady state flow characteristics is presented in Cataño-Lopera et al. (2013). Hereafter the focus of this study is to characterize the unsteady characteristics of the flow structure around the partially buried objects.

**Vortex shedding structure results**

The present case in which the object, SC or TC, reposes upon a deformed bed, displays a much more complex flow structure than results from the interaction of flow passing around a SC upon a flat bed. In the present case, flow structures of diverse scales such as those generated by the object, the scour hole, and the surrounding bedforms add complexity to the flow structure. This is indeed a complex flow case in which diverse 3D vortex flow structures coexist. The complete 3D vortex structure is difficult to discern with the vorticity field, since the vorticity not only includes the rotational field components, but also signals from the boundary generated shear (Smith & Foster 2007). Regions of high boundary shear develop at the sides and top of the object, as well as within the scour hole US of the object. The three vorticity components in the three planes ($xy$, $xz$, and $yz$) are presented with distinctive colors as a first approximation by neglecting the effect of shear generated signals into the vorticity field. Hereafter, rotational zones comprised within the $xy$-, $xz$-, and $yz$-planes are referred to as $z$-vorticity, $y$-vorticity, and $x$-vorticity, respectively, following the right-hand positive convention rule.

The simulations allowed identification of two main flow vortex structures, the upstream HV and the downstream AV systems (Figures 3 and 4). Upstream of the object, a HV forms as a result of the blockage due to the obstacle. Figures 5 and 6 show the main (most significant levels) vorticity contours around the two objects at stationary conditions from the LES simulations. Green represents $y$-vorticity, blue represents $x$-vorticity, and red represents $z$-vorticity. Note that the HV starts as a coherent structure comprised mostly in the $xz$-plane within the scour hole, and progressively transitions to $x$-vorticity cores (blue tubes) by the sides of the object and extends for some distance $DS$ of the object where it loses strength and dissipates as it interacts with the mean flow. The combination of the $y$-vorticity and $x$-vorticity cores gives rise to the HV vortex core behind the object. This HV originates US of the object and progressively helps form the scour hole by entraining sediment as it moves up and down shedding sediment toward the edges and DS of the object. At the same time, a higher intensity $y$-vorticity zone forms above the object (more precisely at the top of the SC and at the leading edge, at $x = −5$ cm, of the TC). The vorticity region extends for about the same distance $DS$ of the two objects, but the vertical extension is larger in the case of the TC than in the SC. Note the presence of $y$-vorticity regions (green) generated at the crest of the surrounding ripples. The resulting interaction among vorticity regions generated by both the object and the ripples gives rise to a much more complex flow structure when compared to the simpler case of an object lying upon a flat bed. In particular, the ripples US of the object disturb the HV around the object, preventing it from displaying a clean and complete regular structure.

Coherent $z$-vorticity structures form at the sides of the object, with higher intensity in the case of the SC than in the TC case. This can be attributed to the sharp edges of the SC compared to the smoother, rounder face of the TC. The $y$-vorticity and the $z$-vorticity structures, along with a smaller contribution from the squeezing in of center convergent $x$-vorticity structures, give rise to the AV. While the AV grows, it disconnects from the shear generation area and is ejected into the exterior flow. However, as the vortex is ejected and mixes with the exterior flow, the root of the shedding AV system remains connected to the generation area at
the top and sides of the object, accompanied by consistently rotating tail x-vortex extensions. At the same time, rotating tail y- and z-vortex extensions remain attached to the object. The $\kappa$-$\varepsilon$ model proved unable to predict the periodic motion associated with the vortex shedding, which results in over-prediction of the length of the recirculation zone, since it does not capture fluid exchange in the near wake, as pointed out by Rodi (1997) and Smith & Foster (2007). Figures 3 and 4 show that the length of the US separation zone is about one object diameter ($D$ and $D_{cb}$ for the SC and TC, respectively) and is located within the scour hole. This separation length, $S$, is smaller than experimental observations in the case of SCs upon a flat bed (Testik et al. 2005). Smith & Foster (2007) showed that the length of the separation zone US of the object tends to constant values of about four and five object diameters for $\kappa$-$\varepsilon$ and LES turbulence models, respectively. In the present study, simulations of the two objects upon a flat immobile bed were also conducted. These simulations revealed that in both cases the length of the separation zone US of the object is about two to three object diameters, which is larger than the simulations for buried conditions. In the case of the recirculation length, $R_0$, defined as the distance between the center of the object and the location where velocity path lines converge in the horizontal $xy$-plane along $y = 0$ cm, is about four times the diameter of the object. This is consistent with the observations of Testik et al. (2005) and Smith & Foster (2007). The latter study showed that for object Reynolds number, $R_{cb}$, larger than 1,500, both the US separation length and the DS recirculation length reached asymptotic constant values. In both cases, LES predicted lengths were generally about 5 to 10% larger than the $\kappa$-$\varepsilon$ predictions.

The LES simulations capture the unsteady dynamics of the most important energetic features of the flow and reveal that the AV regeneration process takes place semi-periodically in a similar manner to vortex shedding in the wake of bridge piers (e.g., Kirkil et al. 2008). The unsteady vortex dynamics along the centerline of the object, $y = 0$ cm, is investigated here by plotting snapshots of contours of $y$-vorticity in Figures 3 and 4 for the two objects.

Upstream of the object, several vortex features are observed. A streak of high vorticity, resulting from the upstream boundary layer (BL) associated with the downflow US of the object, interacts with the main HV existing within the scour hole. The flow inside the HV system is characterized by high levels of energy fluctuations as revealed by the LES simulations (Figures 3 and 4). Such levels are significantly larger than the levels associated with the turbulence of the incoming turbulent BL.

The HV does not have sufficient energy to continue scouring at the equilibrium stage despite the high vorticity. In the SC case, the HV fluctuates and changes size and shape over time, as can be observed at $t = 77.0$ and 77.5 s. Only the LES simulations reveal both positive and negative vorticity regions. Positive and negative vorticity regions refer to counterclockwise and clockwise rotations, respectively. Positive rotation appears as red in Figures 3 and 4, while negative rotations are displayed in blue. In the panel for $t = 77.0$ s, the presence of three separate vortex cores US of the SC are present. The main HV and a smaller one near the bottom of the scour hole (PV2) have positive rotation; the third is a negative rotation vortex attached to the bottom in the US sloping region of the scour hole. In the next panel, for $t = 77.5$ s, the two positive cores, HV and PV2, have merged and the negative vortex (NV1) still remains but is now accompanied momentarily by a second and smaller negative vortex (NV2) at the bottom of the scour hole. At a later time, $t = 78$ s, the newborn NV2 structure disappears while the elongated main HV structure moves. At the same time, NV1 is smashed down by the HV core and divides into two smaller cores as observed at the $t = 79.0$ s panel. Other smaller vortex features appear and disappear intermittently. The main features, however, fluctuate periodically. Notice, for instance, that at a later time, e.g., at $t = 94.5$ s, the two negative vortices are present but the second one is now attached to the US side of the SC. At a later time, the two negative vortex structures, NV1 and NV2, are present but NV2 is now back at the position where it was located at $t = 79.0$ s. At the same time, the main HV moves horizontally and vertically and appears to give birth to a second smaller positive vortex (HV2) towards the top of the SC. Added to the active dynamics of the vortex structures within the scour hole, observe the superposition of a second BL vorticity streak coming from US and interacting with the existing structures. Figure 4 demonstrates that similar vortex structures are revealed and the break ups of the main HV are more dramatic than in the SC case. The main HV breaks...
up into two and three smaller vortex cores as evidenced in panels at times $t = 58.0 \text{s}$ and $t = 60.0 \text{s}$, respectively. This clearly shows the complex unsteady dynamics of the vorticity structures US of the object.

LES simulations show that the AV forms at the top of the object and the regions of higher intensity $y$-vorticity extend for about three object diameters in the DS direction. However, the LES reveals a vortex core more extended in the vertical, as compared to the narrower shape from the $\kappa$-$\varepsilon$ simulation. At the same time, the LES reveals an array of diverse positive and negative rotation vorticity regions, with a much more complex distribution than US of the object. While in the $\kappa$-$\varepsilon$ case, the region of high vorticity extends DS along a relatively steady streak to the point in which the high vorticity dissipates into the exterior flow, the LES picture displays the formation of a high $y$-vorticity vortex (roller) about one diameter DS of the SC. This roller initially forms near the top of the object and grows while moving DS until it breaks up, giving rise to a larger positive rotating vortex that is convected DS by the flow. The breakup is influenced by two mechanisms. The first is by the squeezing action of the incoming $z$-vorticity cores originating from the sides of the object (red contours in Figures 5 and 6) and moving towards the centerline. The second mechanism is the interaction between the roller of positive rotation and negative rotation vortex cores generated above the central sand ridge. These negative rotating cores are associated with bursting phenomena related to turbulent eddies in the near-bed region (Figures 3 and 4). Such cores detach from the bottom and grow in size while rising and merging with others, eventually disrupting the existing positive rotating roller. Meanwhile, other positive bursting vortex cores form near the bed. The conjunction of the two mechanisms results in the roller breakup. The left snapshot panel (top row in Figure 3) shows the released positive vortex in the wake at about one diameter above the SC and about half the TC height (top row of left panel in Figure 4). The subsequent panels in Figure 3 show the full released vortex core migrating DS along with the formation and growth of a new vortex core. The process of roller formation and release continues to happen semi-periodically. A similar situation is observed in the case of the TC in Figure 4.

The Strouhal number, $S_t$, associated with the most energetic frequencies of the released vortices DS of the object is estimated by counting the number of released vortices from the AV system over time. The Strouhal number is given by $S_t = D/U_{Ts}$, where $T_s$ is the period of the vortex shedding oscillation. Note that $D_m$ should be used for the TC case. The results are in very good agreement with the experimental values of 0.41 and 0.36, for the SC and TC, respectively.

The three-dimensional, time-dependent dynamics of the flow are further examined with the help of Figures 5 and 6. The LES simulations give a more complete picture of the unsteadiness of the 3D structures that form around the object. The $\kappa$-$\varepsilon$ RANS-based model presents a sort of quasi-steady state condition of the most significant large-scale coherent turbulence structures associated with the HV formed US of the object, as well as the AV in the wake of the object; while the LES turbulence model allows investigation of the unsteady dynamics of the main coherent vortices. The simulations suggest that during the process of scour hole formation, the large-scale structures detaching from the legs of the main HV transport the entrained sediment particles towards the wake of the object. Such particles tend to deposit once the turbulent eddy structures weaken and mix with the outer flow in the wake region. Other sediment particles are moved towards the centerline of the object, $y = 0 \text{ cm}$, and deposit once the carrier eddies interact with each other and with the small-scale turbulence structures generated near the bed. This leads to the formation of the observed central sand ridges (Figure 2). The sediment particles are lifted mainly by the main HV structure and, to a lesser extent, by the turbulent bursting eddies that occur at the near-bed region. The sweeping motion associated with the vortex structure near the bed may induce significant instantaneous values of bed shear stresses responsible for local sediment entrainment. Peak instantaneous values of the shear velocity are arguably more determinant in sediment entrainment than the mean values.

As mentioned above, the AV vortex forms mainly due to the combination of the most coherent $y$-vorticity and $z$-vorticity structures generated at the top and sides of the object, respectively. The attached boundary layer generated atop and on the sides of the object separates to form the detached...
shear layer (DSL). In Figures 5 and 6, y-vorticity regions are represented in green and x-vorticity regions are represented in red. Once the AV forms, it grows and is carried out by the exterior current. Finally, it becomes unstable and loses energy. At the same time, the ascending x-vorticity tubes coming from the sides of the object squeeze and pinch the AV, and the coherence of the structure is disrupted and loses strength (second row in Figures 5 and 6). Observe that the x-vorticity tubes come from the side and ascend from the scour hole formed at the sides of it. At this point, the front of the AV is released from the structure and a y-vortex core sheds away (second row in Figures 3 and 4). Other coherent structures of smaller scales, compared to those forming the AV, also form at the object boundary and detach from the DSL at frequencies that are dependent on the object Reynolds number. These smaller structures are independent of the shedding frequency associated with the vortex cores released from the main AV. This is clearly observed from the time evolution movies of the present LES simulations, from which the 2D and 3D panel snapshots in Figures 3 through 6 were extracted. The higher the flow and object associated Reynolds numbers, the wider the range of turbulence scales in the flow, which results in the enhancement of shear layer entrainment (Castro & Haque 1988). The formation and evolution of the AV, along with its interaction with the x-vortex tubes (blue in Figures 5 and 6), repeats over time in a semi-periodic manner. The x-vortex tubes forming the legs of the main HV extend and retract in a similar fashion. The legs of the HV first elongate and squeeze in to interact with both the AV and other small-scale vortex structures formed within the DSL, and as a result, all structures lose coherence and break up. The resulting pieces are shed away and the x-tubes shorten, and later the HV system recovers its coherence and the process repeats itself over time. The vortex shedding is responsible in part for the transport and deposition of sediment particles carried into suspension by the HV vortex from the surrounding scour hole. The vorticity on the lateral sides of the objects is oriented mainly in the streamwise direction as indicated by the blue x-vorticity tubes in the left panels of Figures 3 and 4. These vorticity tubes oscillate over time and induce sediment entrainment from the deeper scour hole regions and, at the same time, carry sediment laterally against the slope of the bed. This helps explain the lateral and longitudinal growth of the scour hole during the scour process.

The coherent structures (HV, AV, and detached rollers) carry sediment in the wake of several object diameters DS in the direction in which they are coherent. Once the coherent structures lose intensity, the sediment falls and deposits. The intensity of such coherent structures remains the longest over the centerline of the object and depends strongly on the magnitude of the current. The higher the flow Reynolds number, the farther downstream the vortex shedding extends. In the present case, since the Re is about 80,000 for both objects, the vortex shedding extends over ten object diameters, as shown in Figures 3 and 4. In the far field where the vortex shedding region is small, the bed topography configuration becomes indistinguishable from the region upstream of the object.

Remarks

The present numerical study sheds light on the limitations associated with relying solely on acoustic techniques for flow measurements, e.g., ADVs and ADCPs, when trying to describe flow in the vicinity of solid boundaries. The inherent limitations of the ADV to obtain accurate measurements close to boundaries may lead to drawing erroneous conclusions. Note, for instance, in Figure 4, that in the hole behind the TC, the simulations show a core of negative velocities (blue) very close to the object below the positive one (red) at the top of the TC plus another positive core (red). However, due the lack of detailed measurements very near the bed, only a positive core US of the object away from the bottom is resolved. The two simulated opposite cores observed US of the object indicate the existence of the HV vortex core that rotates counterclockwise. This core is observed in Figures 3 and 4 and is the main cause of sediment extraction from that region, leading to the scour hole. However, after reaching equilibrium conditions, the near-bed velocities lack the strength to induce further deepening of the scour hole.

Observe the complex arrangement of 3D vortex structures in Figures 3 through 6, which is due to the higher Re numbers. In Smith & Foster (2007), the vortex cores are well organized due to the relatively small Re.
CONCLUSIONS

Simulations of the flow structure around two partially buried 3D objects, a SC and a TC, were performed for Reynolds numbers of \( R_e = 86,061 \) and 76,209, respectively. Simulated mean flow velocities were validated against measured velocities using the RANS \( \kappa - \varepsilon \) model. The \( \kappa - \varepsilon \) turbulent model was able to reproduce the mean flow characteristics and mean turbulent intensities observed in the experiments, but failed to resolve the vortex shedding structures in the wake of the object. For this reason, an LES scheme was chosen to investigate the unsteadiness and amplitude of oscillation associated with turbulent structures. For the SC and TC, the vortex shedding was semi-periodic with Strouhal numbers of about 0.35. The length of the wake was in agreement with the extensions of the scouring zone and was similarly predicted with the two turbulence closure schemes, \( \kappa - \varepsilon \) and LES.

The simulations show the rather complex interaction of turbulence structures as revealed mainly by the LES runs. These structures display fluctuating behavior in which bursting and lifting events are observed near the bed. The HV structure behind the object bounces up and down semi-periodically and interacts with the AV generated at the top of the object. However, such interaction is no longer observed at scour equilibrium conditions due mainly to the fact that the HV is deeper inside the scour hole and effectively disconnected from the AV structure.

LESs permitted identifying instantaneous \( y \)-vorticity structures, not revealed by the measurements, responsible for the scour hole US of the object. Such structures are observed during measurements with the object mounted upon both a flat bed and a deformed bed. The up-and-down pulsating ‘hammering’ effect of the vortex structure (with a period \( \sim 3.2 \) s) is evident. This motion is directly connected to the HV structure that is advected DS by the current. Positive \( y \)-vortex cores form atop and DS of the object. The HV extracts sediment that is shed away by the current toward the edges. Counter rotating \( z \)-vorticity structures that form at the lateral sides of the object transport sediment past the object and shed it away towards the centerline. Both \( z \)-vorticity and \( x \)-vorticity cores are responsible for the scouring developed on both sides and DS of the object. The net combination of the core vortices around the object increase the local bed shear stress and enhance sediment erosion, leading to a deepening of the scour hole. Occurrence of \( z \)-vorticity and \( x \)-vorticity on both sides of the object, in addition to the positive \( y \)-vorticity acting from the top, enhance deposition of sediment along the centerline DS of the object, forming the central sand ridge.

Comparison of the flow structures around the SC and TC in the mounted and partially buried cases allow several conclusions to be drawn. First, at the beginning of the experiment, where the object is mounted upon a flat bed, flow intensities are much stronger near the bed level when compared to the case at the mature-equilibrium condition (EMC). This can be attributed to the fact that at the EMC, mean and turbulent intensities are in equilibrium with the entrainment and deposition sediment rates; i.e., at equilibrium the flow is no longer strong enough to make suspension or deposition dominant. During the initial stages of the run, the flow is disturbed in such a way that it is greatly accelerated around the object, and its sediment transport capacity is much higher than in the far field of the object.

Comparison between mounted and partially buried objects showed that the development of the central sand ridge DS of the object disrupts the recirculation zones. This can be readily explained by the ridge preventing the complete development of the vortex structures. An upward directed shedding is observed due to the flow coming from the lower levels at the lateral sides of the object. In the case of a mounted object, the vortex structures have more space to form, while in the case of a partially buried object, the scour holes at US and lateral sides decelerate the flow, allowing vortex structures to form not only in the horizontal \( xy \)-plane but also in the vertical \( xz \)-plane. This helps explain the lateral and longitudinal growth of the scour hole during the scour process.

ACKNOWLEDGEMENTS

This research was originally funded by the Coastal Geophysics Program of the US Office of Naval Research,
Grant N0014-01-1-0337. The work was recently completed thanks to the financial support of the Department of Defense (DOD) Environmental Research Program SERDP (Project Number: MR-2410). The support by both of these programs is gratefully acknowledged. The authors also want to thank the continuous support of the University of Illinois at Urbana-Champaign.

REFERENCES


Huang, W., Yang, Q. & Xiao, H. 2009 CFD modeling of scale effects on turbulence flow and scour around bridge piers. Computers Fluids 38, 1050–1058.


Wilcox, D. C. 2000 Turbulence Modeling for CFD, 2nd edn. DCW Industries, La Cañada Flintridge, CA, USA.

First received 29 May 2016; accepted in revised form 16 August 2016. Available online 17 September 2016