Proceedings of the 39th IAHR World Congress
19–24 June 2022, Granada, Spain
doi://10.3850/IAHR-39WC2521716X2022360

Numerical Simulation of Turbulent Free Surface Flows over Rough Beds

Zhihua Xie(1), Santiago Martelo(2), Thorsten Stoesser(3), Junqiang Xia(4) and Pengzhi Lin(5)

(1) School of Engineering, Cardiff University, Cardiff, UK,
e-mail: zxie@cardiff.ac.uk
(2) School of Engineering, Cardiff University, Cardiff, UK,
e-mail: lopezsm@cardiff.ac.uk
(3) Department of Civil, Environmental & Geomatic Engineering, University College London, London, UK
e-mail: t.stoesser@ucl.ac.uk
(4) State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan, China
e-mail: xiajq@whu.edu.cn
(5) State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, China
e-mail: cvelinpz@126.com

Abstract

The interfacial flows are present in a wide range of flows that are of interest within engineering applications, from the ubiquitous open-channel flow to coastal and marine hydrodynamics. Although many investigations have been undertaken to study such turbulent flow fields, the current understanding of the roughness effect on the turbulent properties of interfacial flows is still limited. A three-dimensional numerical study has been undertaken to investigate turbulent free-surface flows over rough beds. A two-phase flow model has been developed and the large-eddy simulation (LES) approach has been adopted in the study, for which the filtered Navier–Stokes equations are solved and the dynamic Smagorinsky sub-grid model is employed to compute the unresolved scales of turbulence. The governing equations have been discretised using the finite volume method, with the air–water interface being captured using a volume of fluid method and the cut-cell method being implemented to deal with complex geometry in the Cartesian grid. Several numerical examples of turbulent interfacial flow problems are presented to demonstrate the ability of the numerical framework. Detailed mean flow field and turbulence structures are shown and their interaction with the interface are discussed.

Keywords: Free surface flow; Turbulence structure; Large-eddy simulation; Rough beds; Two-phase flow

1. INTRODUCTION

Turbulence occurs in most practical free surface flows. Extensive works have been undertaken to study turbulent flows over rough surfaces (Jiménez, 2004) and, in particular, in open-channels (Nezu and Nakagawa, 1993). However, the current understanding of turbulent free surface flows over rough beds is still limited, especially in shallow water flows, in which the water depth is similar to, or one order of magnitude higher than, the bed roughness.

Many experimental (Nikora et al., 2019) and theoretical (Nikora et al., 2007) studies of rough bed channel and river flows have been undertaken over the past decades, providing useful insights into the mean flow and turbulence statistics for practical engineering applications Most previous numerical model studies over rough beds have been either based on the depth-integrated model, in which simple empirical coefficients, such as Chezy, or Manning coefficient, have been used to link the mean flow variables to the bed roughness, or the Reynolds-averaged Navier-Stokes equations, in which roughness effects have been taken into account using the wall function approach and all of the unsteadiness is averaged out and considered as a part of the turbulence using various approximate methods. In order to overcome the above-mentioned limitations, some direct numerical simulation (DNS) and large-eddy simulation (LES) studies have been undertaken to investigate details of the turbulent flow over rough surfaces. Most previous studies were performed for turbulent flows over well-defined roughness, such as square bars (Cui et al., 2003), wavy bed (Calhoun and Street, 2001), ripples (Zedler and Street, 2001), one layer of spheres (McSherry et al., 2016; Singh et al., 2007), and sand dunes (Xie et al., 2014; Stoesser *et al.*, 2008; Yue et al., 2006). For turbulent open-channel flows over a gravel bed, a porosity algorithm has been used by Hardy et al. (2007) and a roughness geometry function together with forcing terms in the momentum equations has been used by Stoesser (2010).

The objective of this study is to use the LES approach to investigate turbulent shallow water flows over rough beds, with the cut cell method being implemented in a Cartesian grid to deal with the complex topography. The model predicted flow field is presented and compared with available experimental measurements. The LES acts as a complementary approach to experimental investigations, providing a more detailed space-time resolution of the flow field to gain further insight into the turbulent flow dynamics.

2. MATHEMATICAL MODEL

The large-eddy simulation (LES) approach is adopted in this study, and the governing equations used for incompressible two-phase flow are based on the filtered Navier–Stokes equations, given as

$$\nabla \cdot \bar{\boldsymbol{u}} = 0 , \quad [1]$$

$$\frac{\partial(\rho\bar{\boldsymbol{u}})}{\partial t} + \nabla \cdot (\rho\bar{\boldsymbol{u}} \otimes \bar{\boldsymbol{u}}) = -\nabla\bar{p} + \nabla \cdot \left[\mu\left(\nabla\bar{\boldsymbol{u}} + \nabla^{\mathrm{T}}\bar{\boldsymbol{u}}\right)\right] + \rho\boldsymbol{g} + \nabla\boldsymbol{\tau}^{\mathrm{sgs}} , \quad [2]$$

where the overbar $\bar{\cdot}$ denotes the spatial filtering over the grid in Cartesian coordinates (x , y and z , meaning the streamwise, vertical, and spanwise directions, respectively). $\bar{\boldsymbol{u}} = (\bar{u}, \bar{v}, \bar{w})$ is the filtered velocity vector, t is time, $\bar{p}$ is the filtered pressure, $\boldsymbol{g}$ is the gravitational acceleration vector, ρ is density and μ is the dynamic viscosity of the fluid.

The term $\boldsymbol{\tau}^{\mathrm{sgs}} = \rho\left(\bar{\boldsymbol{u}}\bar{\boldsymbol{u}} - \overline{\boldsymbol{u}\boldsymbol{u}}\right)$ is the sub-grid scale (SGS) stress tensor and the anisotropic part of the SGS term is modelled by an eddy-viscosity model of the form (Lily, 1992):

$$\boldsymbol{\tau}^{\mathrm{sgs}} - \frac{1}{3}\mathrm{trace}(\boldsymbol{\tau}^{\mathrm{sgs}})\boldsymbol{I} = 2u_t\bar{\boldsymbol{S}} , \quad [3]$$

where $\boldsymbol{I}$ is the unit tensor and $\bar{\boldsymbol{S}}$ is the strain rate tensor of the resolved field. u_t is the turbulent eddy viscosity defined as:

$$u_t = \rho C_d \bar{\Delta}^2 \left|\bar{\boldsymbol{S}}\right| , \text{ and } \left|\bar{\boldsymbol{S}}\right| = \sqrt{2\bar{\boldsymbol{S}}\bar{\boldsymbol{S}}} \quad [4]$$

With the cut-off length scale $\bar{\Delta} = (\Delta x \Delta y \Delta z)^{1/3}$ and the coefficient C_d . In this study, the dynamic subgrid model (Lily, 1992; Germano et al., 1991) is used to determine the model coefficient C_d .

The momentum equation is closed with the constitutive relations for the density and dynamic viscosity of the fluid is given by:

$$\rho = F\rho^{\mathrm{w}} + (1-F)\rho^{\mathrm{a}} , \quad [5]$$

$$\mu = F\mu^{\mathrm{w}} + (1-F)\mu^{\mathrm{a}} , \quad [6]$$

where the superscripts ‘w’ and ‘a’ denote water and air, respectively. F is the volume fraction defined as:

$$F = \begin{cases} 1, & \text{if only water is present;} \\ 0, & \text{if only air is present.} \end{cases} \quad [7]$$

The air-water interface is then within the cells where $0 < F < 1$. A particle on the surface stays on the surface and the volume fraction F has a zero material derivative:

$$\frac{\mathrm{d}F}{\mathrm{d}t} = \frac{\partial F}{\partial t} + \bar{\boldsymbol{u}} \cdot \nabla F = 0 . \quad [8]$$

These equations complete the mathematical description of the two-phase flow model.

3. NUMERICAL METHOD

For the sake of brevity, only a brief description of the numerical method is presented here, and more details of the numerical method for the in-house multiphase flow solver (Xdolphin3D) can be found in previous works (Xie and Stoesser, 2020; Xie, 2015). In this study, the governing equations are discretised using the finite volume method on a staggered Cartesian grid. The advection terms are discretised by a high-resolution scheme, which combines the high order accuracy with monotonicity, whereas the gradients in pressure and diffusion terms are obtained by central difference schemes. In order to deal with complex geometries in Cartesian grids, the cut cell method in 3D (Xie and Stoesser, 2020) was utilized in the finite volume discretisation. The advective and diffusive fluxes at cell faces, as well as cell volume, are modified in cut cells. The SIMPLE algorithm is employed in the present study for the pressure-velocity coupling and a second-order backward finite difference discretisation is used for the time derivative, which leads to an implicit scheme for the governing equations. In this study, the air-water interface is captured by a high-resolution VOF scheme CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes) (Ubbink, 1997), which does not need to reconstruct the interface after each time step. The multiphase flow code Xdolphin3D has already been extensively verified and validated through numerous test cases for breaking waves (Xie, 2015), wave-structure interaction (Xie et al., 2020), and LES studies of free surface flow over rough surfaces (Xie et al., 2022; Xie et al., 2014; Xie et al., 2013).

4. RESULTS AND DISCUSSION

The large-scale coherent structures generated over rough bed play an essential role in the interaction between the bed and free surface, sediment transport, and bedform evolution in open-channel flows. In order to illustrate the coherent vortical structure developed in the flow over the rough bed, the λ2 method (Jeong and Hussain, 1995) was used in the present study to identify vortex cores, based on the second invariant of the velocity gradient tensor.

Figure 1 shows an example of the snapshot of the water surface and vortical structures in the turbulent free-surface flow over three-dimensional dunes, in which the vortical structures are coloured by the vertical distance and the water surface is coloured by the vertical distance to the mean water depth. Strong turbulence structures are generated in the separation zone and the generated hairpin vortices are advected downstream along the dune, with complex temporal and spatial interactions occurring between these vortical structures.

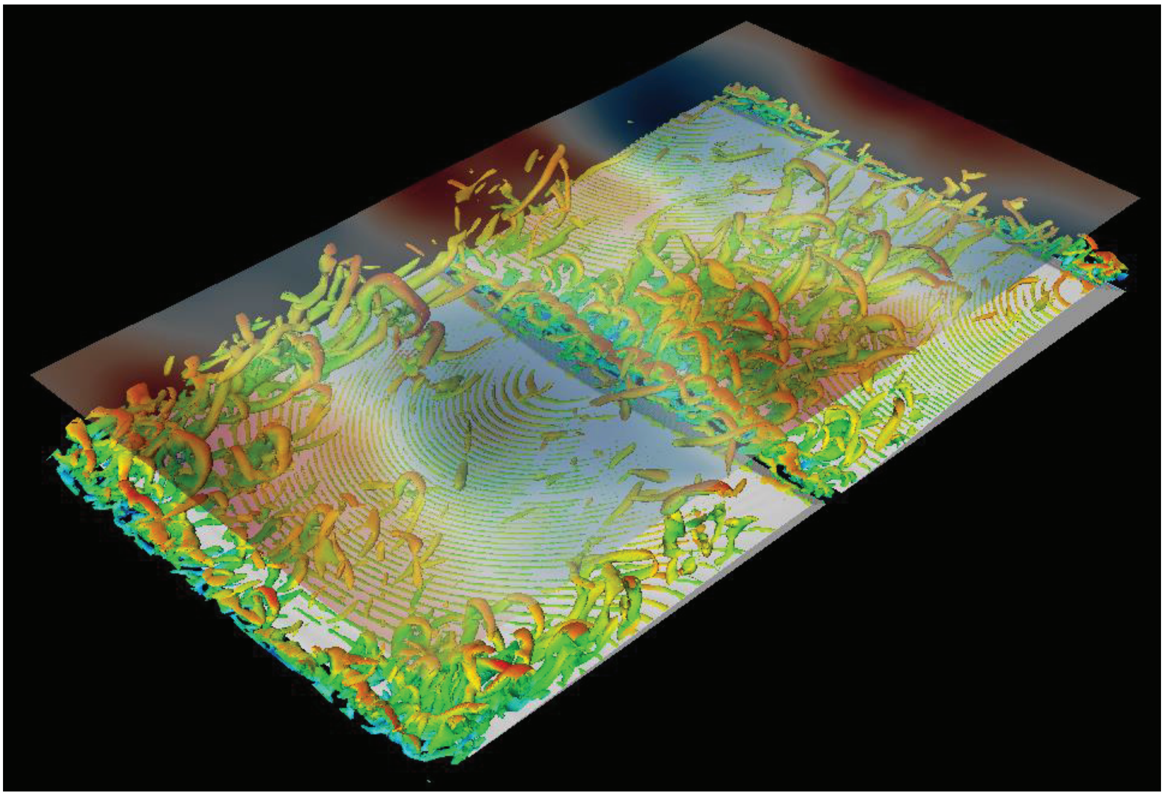

Figure 1. The air-water interface and coherent structures in turbulent free-surface flow over three-dimensional dunes.

Figure 2 shows an example of the snapshot of the water surface and vortical structures in the turbulent free-surface flow over a gravel bed. It can be observed that vortical structures are generated in the near-bed region and are advected downstream, with complex interactions with the bed underneath and the free surface above.

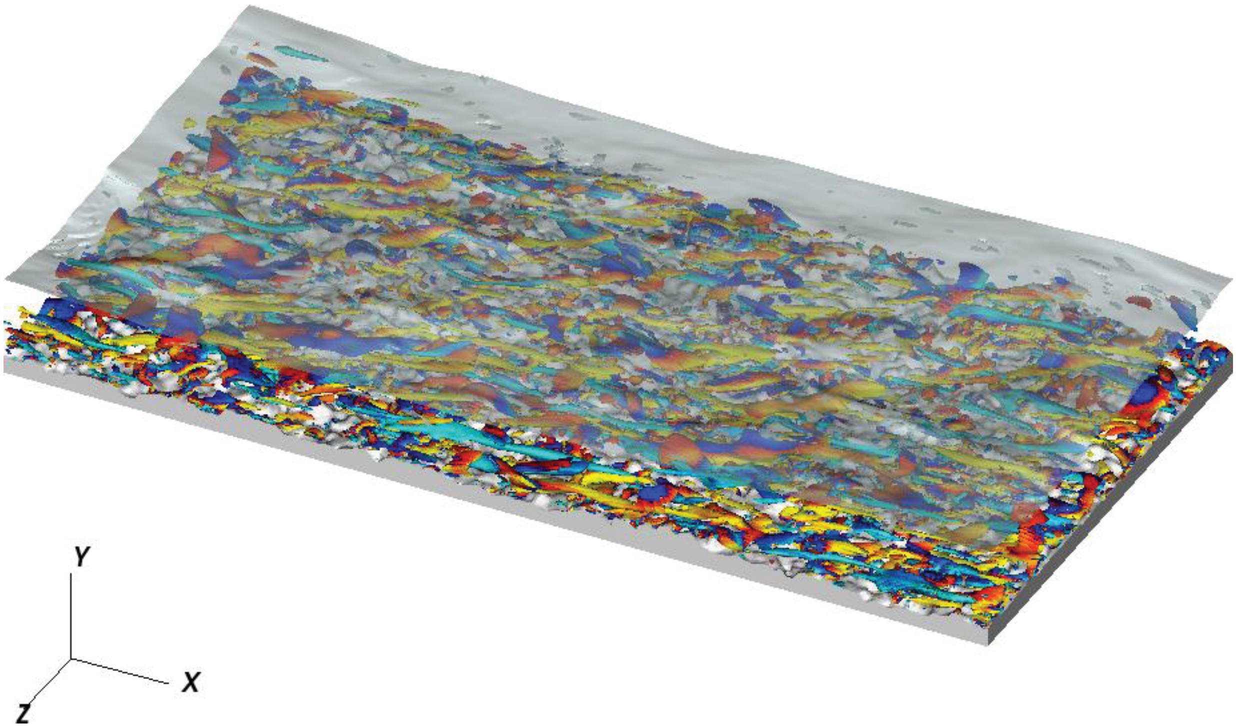

Figure 2. The air-water interface and coherent structures in turbulent free-surface flow over a gravel bed.

5. CONCLUSIONS

In this study, a large-eddy simulation of turbulent free-surface flow over rough beds has been presented. The Cartesian cut-cell method has been used to represent the complex geometry in the numerical model. In addition, the deformable free-surface has been captured using the VOF method, which is different from the rigid lid approximation being used in other open-channel flow simulations.

More detailed results of the free-surface dynamics and turbulence statistics will be presented at the conference. In addition, numerical results of coherent structures for various rough surfaces will be shown and discussed.

The study demonstrates the capability of the LES code Xdolphin3D to provide reliable detailed turbulent flow characteristics, which is necessary in order to obtain a better understanding of the turbulent flow dynamics in fluvial systems.

6. ACKNOWLEDGEMENTS

We would like to thank the EPSRC projects (number EP/R022135/1, EP/S016376/1) for helping to fund this work. This work is also supported by the Royal Society Newton Advanced Fellowship (NAF/R1/201156) and the Open Fund of the State Key Laboratory of Hydraulics and Mountain River Engineering at Sichuan University (SKHL1904). The simulations were performed at the Supercomputing Wales clusters, which are greatly acknowledged.

7. REFERENCES

Calhoun, R.J. and Street, R.L. (2001). Turbulent flow over a wavy surface: neutral case. *Journal of Geophysical Research-Oceans*, 106, 9277–9293.

Cui, J., Patel, V.C. and Lin, C.L. (2003). Large-eddy simulation of turbulent flow in a channel with rib roughness. *International Journal of Heat and Fluid Flow*, 24(3), 372–388.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H. (1991). A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A*, 3(7), 1760–1765.

Hardy, R.J., Lane, S.N, Ferguson, R.I. and Parsons, D.R. (2007). Emergence of coherent flow structures over a gravel surface: a numerical experiment. *Water Resources Research*, 43, W03422.

Jeong, J. and Hussain, F. (1995). On the identification of a vortex. *Journal of Fluid Mechanics*, 285, 69–94.

Jiménez, J. (2004). Turbulent flows over rough walls. *Annual Review of Fluid Mechanics*, 36, 173–196.

Lilly, D.K. (1992). A proposed modification of Germano-subgrid-scale closure method. *Physics of Fluids A*, 4(3), 633–635.

McSherry, R., Chua, K., Stoesser, T., & Falconer, R.A. (2016). Large eddy simulations of rough bed open channel flow with low submergence and free surface tracking. In *River flow 2016* (pp. 85–90). Taylor and Francis.

Nezu, I. and Nakagawu, H. (1993). *Turbulence in Open-Channel Flows*, International Association for Hydraulic Research Monograph Series. A.A. Balkema, Rotterdam.

Nikora, V., McEwan, I., McLean, S., Coleman, S., Pokrajac, D. and Walters, R. (2007). Double-averaging concept for rough-bed open-channel and overland flows: theoretical background. *ASCE Journal of Hydraulic Engineering*, 133(8), 873–883.

Nikora, V., Stoesser, T., Cameron, S.M., Stewart, M., Papadopoulos, K., Ouro, P., McSherry, R., Zampiron, A.,
Marusic, I., & Falconer, R.A. (2019). Friction factor decomposition for rough-wall flows: Theoretical background and application to open-channel flows. *Journal of Fluid Mechanics*, 872, 626–664.

Singh, K.M., Sandham, N.D. and Williams, J.J.R. (2007). Numerical simulation of flow over a rough bed. *ASCE Journal of Hydraulic Engineering*, 133(4), 386–398.

Stoesser, T. (2010). Physically realistic roughness closure scheme to simulate turbulent channel flow over rough beds with the framework of LES. *ASCE Journal of Hydraulic Engineering*, 136(10), 812–819.

Stoesser, T., Braun, C., Garcia-Villalba, M. and Rodi, W. (2008). Turbulence structures in flow over two-dimensional dunes. *ASCE Journal of Hydraulic Engineering*, 134(1), 42–55.

Ubbink, O. (1997). Numerical prediction of two fluid systems with sharp interfaces, PhD thesis, Imperial College of Science, Technology and Medicine.

Xie, Z. (2015). A two-phase flow model for three-dimensional breaking waves over complex topography. *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 471, 20150101.

Xie, Z., Lin, B., Falconer, R.A., and Maddux, T.B. (2013). Large-eddy simulation of turbulent open-channel flow over three-dimensional dunes. *Journal of Hydraulic Research*, 51(5), 494 – 505.

Xie, Z., Lin, B., and Falconer, R.A. (2014). Turbulence characteristics in free-surface flow over two-dimensional dunes. *Journal of Hydro-Environmental Research*, 8(3), 200 – 209.

Xie, Z., and Stoesser, T. (2020). A three-dimensional Cartesian cut-cell/volume-of-fluid method for two-phase flows with moving bodies. *Journal of Computational Physics*, 416, 109536.

Xie, Z., Stoesser, T., Yan, S., Ma, Q., and Lin, P. (2020). A Cartesian cut-cell based multiphase flow model for large-eddy simulation of three-dimensional wave-structure interaction. Computers & Fluids 213, 104747.

Xie, Z., Lin, B., Falconer, R.A., Nichols, A.N. Tait, S., and Horoshenkov, K. (2022). Large-eddy simulation of turbulent free surface flow over a gravel bed. *Journal of Hydraulic Research*, in press.

Yue, W.S., Lin, C.L. and Patel, V.C. (2006). Large-eddy simulation of turbulent flow over a fixed two-dimensional dune. *ASCE Journal of Hydraulic Engineering*, 132(7), 643–651.

Zedler, E.A. and Street, R.L., 2001. Large-eddy simulation of sediment transport: currents over ripples. *ASCE Journal of Hydraulic Engineering*, 127(6), 444–452.

