HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Conference Paper, Published Version

Xie, Zhihua; Lin, Binliang; Falconer, Roger A.; Nichols, Andrew

A Cartesian Grid Method for Shallow Waterflows over a Gravel Bed

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Kuratorium für Forschung im Küsteningenieurwesen (KFKI)

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/109745

Vorgeschlagene Zitierweise/Suggested citation:
Xie, Zhihua; Lin, Binliang; Falconer, Roger A.; Nichols, Andrew (2012): A Cartesian Grid Method for Shallow Waterflows over a Gravel Bed. In: Hagen, S.; Chopra, M.; Madani, K.; Medeiros, S.; Wang, D. (Hg.): ICHE 2012. Proceedings of the 10th International Conference on Hydroscience & Engineering, November 4-8, 2012, Orlando, USA.

Standardnutzungsbedingungen/Terms of Use:

Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.

Proceedings of the 10th Intl. Conf.on Hydroscience & Engineering, Nov. 4-7, 2012, Orlando, Florida, U.S.A.

A CARTESIAN GRID METHOD FOR SHALLOW WATER FLOWS OVER A GRAVEL BED

Zhihua Xie[1], Binliang Lin[2], Roger A. Falconer[3], Andrew Nichols[4]

ABSTRACT

Turbulence in open-channel flow over a rough bed is very complex. Although many investigations have been undertaken, the current understanding of the roughness effect on the turbulent properties is still limited. This paper presents a large-eddy simulation study of turbulent open-channel flow over a gravel bed, whose detailed high-resolution surface elevation data were obtained from a laser displacement sensor in a laboratory. The filtered Navier-Stokes equations have been discredited using the finite volume method, with the dynamic sub-grid model being employed for the unresolved scales of turbulence. In order to accurately represent the gravel bed, the partial cell treatment has been implemented in a Cartesian grid to deal with the bed topography. The model has been used to predict the mean streamwise velocities, turbulent intensities, and Reynolds shear stresses, and the results have been compared with the corresponding particle image velocimetry measurements.

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 open-channel 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 studies of gravel-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 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 and large-eddy simulation (LES) studies have been undertaken to investigate details of the turbulent flow over rough surfaces. However, 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 (Singh *et al*., 2007), and sand dunes (Yue *et*

---

[1] Research Associate, School of Engineering, Cardiff University, CF24 3AA, UK (zxie@cardiff.ac.uk)

[2] Professor, State Key Laboratory of Hydroscience and Engineering, Tsinghua University, 100084, China; School of Engineering, Cardiff University, CF24 3AA, UK (linbl@tsinghua.edu.cn)

[3] Professor, School of Engineering, Cardiff University, CF24 3AA, UK (falconerra@cardiff.ac.uk)

[4] Research Assistant, School of Engineering Design and Technology, University of Bradford, BD7 1DP, UK (a.nichols2@bradford.ac.uk)

al., 2006; Stoesser *et al.*, 2008). 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 a gravel bed, with the partial cell treatment being implemented in a Cartesian grid to deal with the complex topography. Model predicted mean velocity and turbulence statistics are presented and compared with the experimental measurements obtained at the University of Bradford. The LES acts as a complementary approach to experimental investigations, providing more detailed space-time resolution of the flow field to gain further insight to the turbulent flow dynamics.

2. EXPERIMENTS

A series of experiments was conducted at the University of Bradford in which the behavior of the water surface was altered by adjusting the general flow conditions of a range of shallow flows over a rough, sediment boundary. The experiments were carried out in a 12 m long, sloping rectangular flume. The flume is 0.459 m wide and it was set to a fixed slope of 0.004. Volumetric flow rates of up to 0.04 m^3/s were used in the experiments. The flume (Figure 1(a)) had a well-mixed gravel bed placed in the base of the flume which was composed of washed river gravel with the density of ρ_g = 2600 kg/m^3 and grain size D_{50} = 4.4 mm. The gravel bed was scraped to a uniform level and so had no significant topographical features. The bed surface roughness was measured at the test section (Figure 1(b)) of the flume using a laser displacement sensor attached to a computer controlled scanning frame. During these tests the flow structures were measured using particle image velocimetry (PIV) in the flume.

(a) (b)

Figure 1 Hydraulic flume (a) and test section (b).

3. NUMERICAL METHOD

The large-eddy simulation approach was adopted in this study, and the filtered Navier-Stokes equations were discredited using the finite volume method on a staggered Cartesian grid. The sub-grid scale stresses appearing in the filtered Navier-Stokes equations were modeled using the dynamic sub-grid model (Germano *et al.*, 1991; Lilly, 1992). The advection terms were discredited by a high-resolution scheme (Hirsch, 2007), which combined the high order accuracy with monotonicity, whereas the gradients in the pressure and diffusion terms were obtained using the central difference schemes. The SIMPLE algorithm was employed for the pressure-velocity coupling and the second-order Gear's method was used for the time derivative, which led to an implicit solution for the governing equations. The code was parallelized using MPI and a domain decomposition technique.

To deal with complex topographies in engineering applications, overlapping grids, boundary-fitted grids, and unstructured grids can be used. These methods provide great flexibility to conform onto complex stationary or moving boundaries. However, the programming of these methods can be complicated and generating such a grid is usually very cumbersome (Mittal and Iaccarino, 2005). Numerical methods based on a Cartesian grid, which can simulate flow with complex topography on Cartesian grids, avoid these problems. Two of the most popular methods are the immersed boundary method (Mittal and Iaccarino, 2005) and the Cartesian cut cell method (Ingram *et al.*, 2003). The primary advantage of a Cartesian grid method is that only moderate modification of the program on Cartesian grids is needed to account for a complex topography. A Cartesian grid method also has the advantage of being simple to generate, particularly with moving boundary problems, due to the use of stationary, non-deforming grids. However, the drawback of this method is that implementing boundary conditions is not straightforward. For LES studies of turbulent flow over a gravel bed, in the present study the partial cell treatment developed for 2D by Torrey *et al.* (1985) has been extended to 3D and utilized in the finite volume discretisation, in which the advective and diffusive fluxes at cell faces, as well as the cell volume, have been modified in cut cells.

4. COMPUTATIONAL SETUP

The computational model was set up to replicate the flows in the laboratory model studies mentioned above, in which detailed measurements of turbulent open-channel flow over a gravel bed in a laboratory flume were available. The schematic view of the computational domain is shown in Figure 2, where the origin is located at the mean gravel bed elevation at the upstream section along the central plane. A section of the gravel bed with a length L_x = 0.4 m and a width L_z = 0.2 m was selected from the test section in the experiment. The case with flow depth h = 0.07 m (relative to the mean bed elevation) was selected to investigate the roughness effects in open-channel flows, with the ratio of the water depth to the maximum gravel height, i.e. *h/H,* being 5. The corresponding Reynolds and Froude numbers, based on the mean bulk flow velocity $U_{\text{bulk}} \approx$ 0.5 m/s and the water depth h , were Re ≈39, 000 and Fr ≈ 0.6 , respectively. The computational domain of Lx × h × Lz was discredited using a uniform grid of 256 × 128 × 128 points in the streamwise, vertical, and span wise directions, respectively. Periodic boundary conditions were used in the streamwise and span wise directions, whereas the free surface was modeled as a rigid lid (i.e. a free-slip boundary condition), which has been successfully used in previous LES studies of turbulent open-channel flows (e.g. Zedler and Street, 2001; Yue *et al.*, 2006; Stoesser *et al.*, 2008).

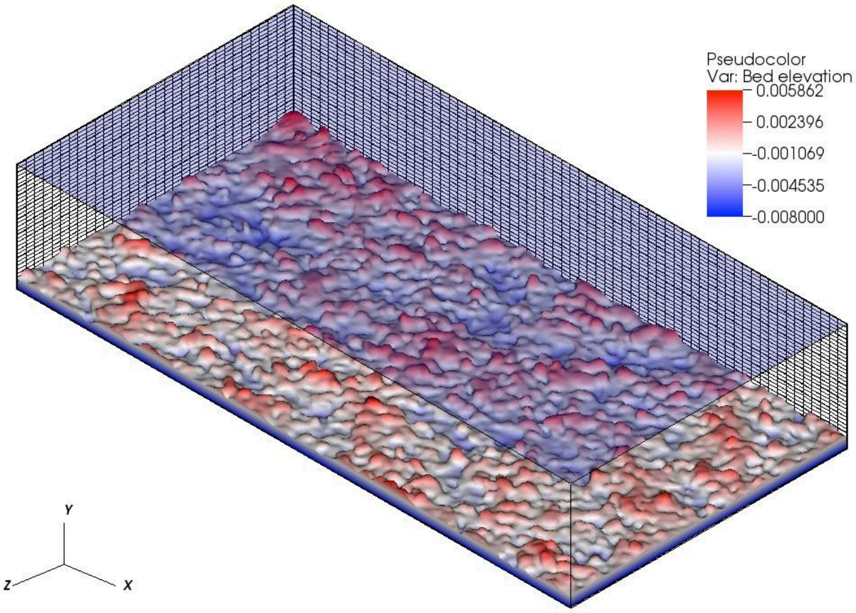

Figure 2 Schematic view of the computational domain with gravel bed elevation given in metres.

5. RESULTS AND DISCUSSION

Because the flow is spatially heterogeneous, the double-averaging (in both time and space) methodology (Nikora *et al,* 2007) is employed to analyze the results. Figure 3 shows the predicted vertical profiles of the double-averaged streamwise velocity ($\langle U \rangle$), turbulence intensities ($\langle u' \rangle$ and $\langle v' \rangle$) and Reynolds shear stress ($\langle -u'v' \rangle$), along with corresponding experimental measurements.

It is shown from Figure 3 that the mean velocity $\langle U \rangle$ exhibits a linear profile below the maximum gravel bed elevation (denoted as dotted lines) in the near-bed region and a logarithmic elevation. As the velocity within the roughness layer is very difficult to measure, the LES can be used as an additional tool to study the details for the near-bed flow.

The streamwise turbulence intensity attains its peak value just below the maximum bed elevation and decreases towards both the free surface and the bed. The $\langle u' \rangle$ is slightly over-predicted by the present model near the maximum bed elevation but a better result is obtained towards the free surface. The vertical turbulence intensity increases from the bottom to a height well predicted in the lower region of the flow and under-predicted when approaching the free surface. This is attributed to the rigid lid approximation used in the LEW model and the corresponding limitation in predicting the vertical velocities. The peak value of the Reynolds shear stress is found at a short distance about the maximum bed elevation and similar trends are observed between the predicted and measured results. However the s $\langle -u'v' \rangle$ is over-predicted by the present model. This discrepancy might be attributed to the fact that the periodic boundary conditions used in the simulation were different from the actual flow condition in the experiment, and which could have enhanced the turbulence levels in the simulations. Overall, a reasonably good agreement between the LES and PIV results is obtained for the turbulence statistics.

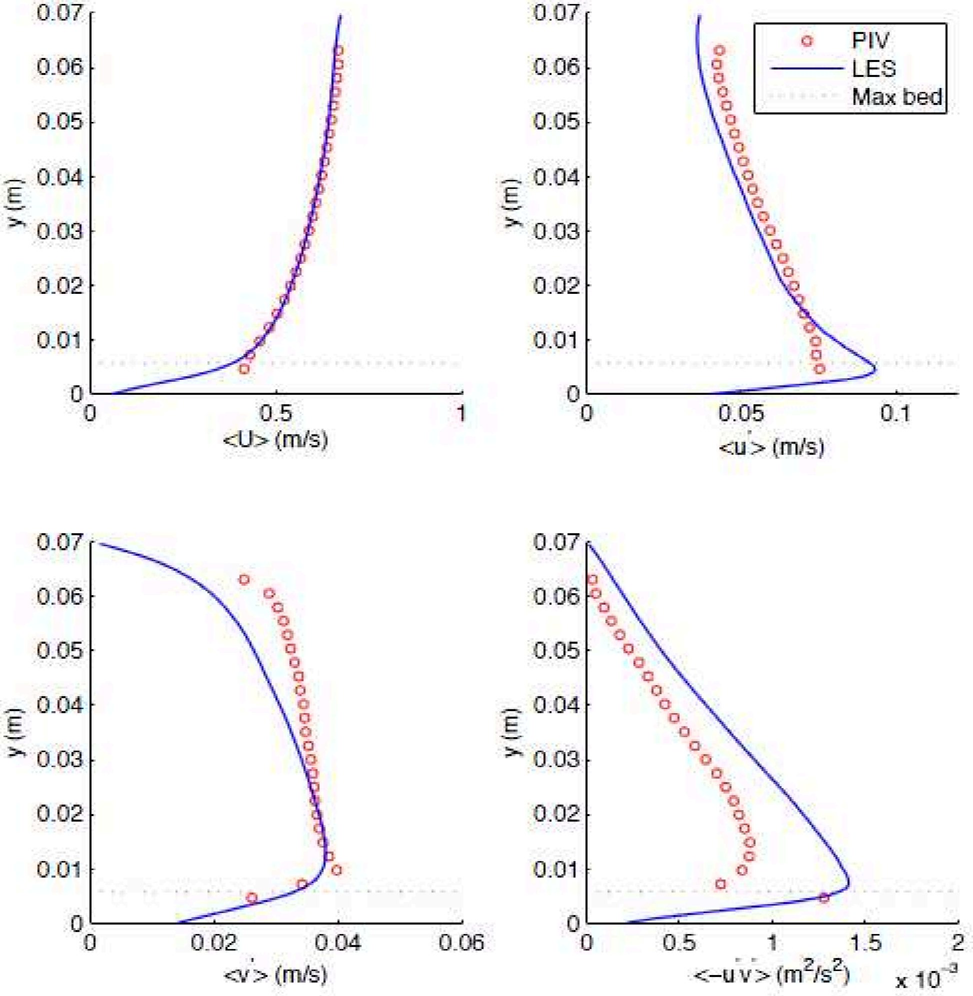

Figure 3 Comparison of predicted and measured results for the mean velocity, turbulence intensities and Reynolds shear stress.

6. CONCLUSIONS

In this paper a large-eddy simulation study of turbulent flow over a gravel bed has been presented, in which a generally good agreement between the model predictions and experimental measurements was obtained. The study demonstrates the capability of the Cartesian grid method used in the present LES model in providing reliable detailed mean velocity and turbulence characteristics along the water depth, which is necessary in order to obtain a better understanding of turbulent flow dynamics in rough open-channel flows.

ACKNOWLEDGEMENTS

The research was supported by the UK Engineering and Physical Sciences Research Council

(EP/G014264/1). This work was performed using the computational facilities of the Advanced Research Computing at Cardiff (ARCCA) Division, Cardiff University.

REFERENCES

Calhoun, R.J., and Street, R.L. (2001) “Turbulent Flow over a Wavy Surface: Neutral Case”, Journal of Geophysical Research-Oceans, Vol. 106, pp. 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, Vol. 24, No. 3, pp. 372-388.

Germano, M., Piomelli, U., Moin, P., and Cabot, W.H. (1991) “A Dynamic Subgrid-Scale Eddy Viscosity Model”, Physics of Fluids A, Vol. 3. No. 7, pp. 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, Vol. 43, W03422.

Hirsch, C. (2007) Numerical Computation Of Internal And External Flows Introduction To The Fundamentals Of CFD, New Edition, Butterworth-Heinemann, Oxford.

Ingram, D.M., Causon, D.M., and Mingham, C.G. (2003) “Developments in Cartesian Cut Cell Methods”, Mathematics and Computers in Simulation, Vol. 61, No. 3-6, pp. 561-572. Jiménez, J. (2004) “Turbulent Flows over Rough Walls”, Annual Review of Fluid Mechanics, Vol. 36, pp. 173-196.

Lilly, D.K. (1992) “A Proposed Modification of the Germano-Subgrid-Scale Closure Method”, Physics of Fluids A, Vol. 4, No. 3, pp. 633-635.

Mittal, R., and Iaccarino, G. (2005) “Immersed Boundary Methods”, Annual Review of Fluid Mechanics, Vol. 37, pp. 239-261.

Nezu, I., and Nakagawa, H. (1993) Turbulence in Open-Channel Flows, International Association of 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”, Journal of Hydraulic Engineering, Vol. 133, No. 8, pp. 873-883.

Singh, K.M., Sandham, N.D., and Williams, J.J.R. (2007) “Numerical Simulation of Flow over a Rough Bed”, Journal of Hydraulic Engineering, Vol. 133, No. 4, pp. 386-398.

Stoesser, T. (2010) “Physically Realistic Roughness Closure Scheme to Simulate Turbulent Channel Flow over Rough Beds with the Framework of LES”, Journal of Hydraulic Engineering, Vol. 136, No. 10, pp. 812-819.

Stoesser, T., Braun, C., Garcia-Villalba, M., and Rodi, W. (2008) “Turbulence Structures in Flow over Two-Dimensional Dunes”, Journal of Hydraulic Engineering, Vol. 134, No. 1, pp. 42-55.

Torrey, M.D., Cloutman, L.D., Mjolsness, R.C., and Hirt, C.W. (1985) “NASA-VOF2D: a Computer Program for Incompressible Flows with Free Surfaces”, Tech. Rep. LA-10612-MS, Los Alamos Scientific Laboratory.

Yue, W.S., Lin, C.L., and Patel, V.C. (2006) “Large-Eddy Simulation of Turbulent Flow over a Fixed Two-Dimensional Dune”, Journal of Hydraulic Engineering, Vol. 132, No. 7, pp. 643-651.

Zedler, E.A., and Street, R.L. (2001) “Large-Eddy Simulation of Sediment Transport: Currents over Ripples”, Journal of Hydraulic Engineering, Vol. 127, No. 6, pp. 444-452.