15th International Symposium on Flow Visualization
June 25-28, 2012, Minsk, Belarus

NEW METHOD FOR VISUALIZING LARGE SCALE TURBULENCE STRUCTURE BY TRACING SHEET OF PSEUDO-TRACER CLOUD

I. FUJITA[1,c], T. OKANISHI[1]

[1]Department of Civil Engineering, Kobe University, 1-1 Rokkodai, Nada, Kobe 657-8501,Japan

[c]Corresponding author: Tel.: +81788036439; Fax: +81788036439; Email: ifujita@kobe-u.ac.jp

KEYWORDS:

Main subjects: computational fluid mechanics, flow visualization
Fluid: open-channel flow, flow with gravel bed
Visualization method(s): particle tracing
Other keywords: LES, rough wall turbulence

ABSTRACT: A new idea of visualizing turbulence structure by tracing a cloud of pseudo-particles repeatedly placed in a horizontal sheet is proposed. The method was applied to rough wall open-channel flows simulated by a large eddy simulation (LES) model. The water surface was expressed by a density function, thereby air flow phase as well as water flow phase was calculated in the simulation. Two types of rough wall boundaries were considered with natural gravels of different diameters. The bathymetries composed of these particles were reproduced by an immersed boundary method (IBM). In the new method, a large number of pseudo-particles are firstly generated in a horizontal sheet moving with their local flow directions. By repeating the generation of pseudo-particles at the same plane, a large scale structure within the turbulent flow can be visualized as a topographic image of the particles. The resultant figure displays a flow structure generated by a number of small fluid volumes passing upward through the horizontal plane. The application of the method to rough wall turbulent flows revealed a new aspect of flow structure generated by roughness elements.

INTRODUCTION

The simulation and experimental techniques for open-channel turbulent flows have made a significant progress in the last decades (Naot et al. 1996, Jimenez 2004, Bates et al. 2005). In particular, the turbulence properties with respect to hydraulically smooth wall beds have been investigated intensively so far and summarized favorably by Nezu and Nakagawa (1993). At the same time, the understanding of rough wall turbulence in open channel flows has been a major theme in the past several decades from the view point of the river engineering (Grass 1971, Graf and Altinakar 1998, Nikora et al. 2001). Though most of such researches were conducted experimentally by measuring detailed flow fields, numerical simulation models capable to treat rough wall bed that accompanies large-scale structures have been developed recently by using a model of large eddy simulation (Cui et al. 2003, Xiaohui and Li 2002). One of the specific features of rough wall turbulence, in addition to the different properties of flow resistance, is that the water surface fluctuation is relatively larger than that for smooth wall turbulent flows. One of the causes of such a fluctuation can be the impingement of large-scale vortices against a water surface; however, the generation mechanism of such surface features has not been fully understood. In the other aspect, it has been recognized recently that water surface features thus generated are advected with the surface flow in a certain condition and can be used as non-intrusive river flow measurements (Fujita et al. 2007, Muste et al. 2008). The transverse surface velocity distributions can also be used to estimate river flow discharges within an error of practical uncertainty. In any case, the source of all these aspects can be the generation of turbulent vortices generated at rough wall beds. For investigating such features of rough wall turbulence in open-channel flows, we developed a numerical simulation model based on a large eddy simulation that is capable to treat water surface fluctuations by introducing a density function. The model is thus a two phase model which calculates air and liquid phases at the same time. The detailed features of rough wall boundary can be introduced to the simulation directly via the information of digital elevation data of the boundary. The problem related to the three dimensional simulation of turbulent flow fields is how to visualize the large-scale structure properly to be able to understand the specific flow properties. For that purpose, we devised a new idea to visualize an organized vortical structure in a horizontal plane by introducing a cloud of pseudo particles that move with local velocity fields.

NUMERICAL SIMULATION MODEL

The numerical simulation is based on the large eddy simulation model in a Cartesian grid system. The fundamental equations used after the spatial filtering is as follows:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\overline{u}_i^{n+1} - \overline{u}_i^{n}}{\Delta t} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} - \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial}{\partial x_i}\left\{ \nu_t \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \right\} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j^2} + f_i \tag{2}$$

Here, x_i is the coordinate in the i-th direction (i=1,2,3), $\overline{u_i}$ is the grid scale velocity in the i-th direction, ν is the kinematic viscosity, ν_t is the eddy viscosity and f_i is the grid-scale external force in the i-th direction. It should be noted that the unsteady term in the filtered Navier-Stokes equation is discretized by a forward differencing.

In rough wall conditions, the bottom boundary will be composed of artificial roughness elements or gravel particles. For representing such complex bottom conditions in the Cartesian coordinates, we introduced the immersed boundary (IB) method proposed by Fadln et al. (2000), the direct forcing method, into the LES algorithm. The IB method represents a wall boundary by adding an external force into the fundamental equations. Here, the external force f has the following form,

$$f_i = -RHS + \frac{U_i^{n+1} - \overline{u}_i^{n}}{\Delta t} \tag{3}$$

where *RHS* stands for the summation from the first to the fourth terms in the right hand side of the equation(2). The variable U_i^{n+1} is an interpolated velocity vector by using values near the wall grids so that the velocity at the boundary becomes zero. This term is set at zero within the wall grids. In the actual implementation of IB method to the model, an algorithm was developed so that a digital elevation data (DED) can be directly linked to the main solver of the LES program.

In the LES application to complicated boundary configurations, the conventional Smagorinsky's model is difficult to apply, because the damping function used in the model is a function of the distance from the boundary and it is difficult to measure such a distance for a complicated topography. The dynamic SGS model developed by Germano et al. (1991) can solve this problem to a certain extent, however this model tends to be unstable in some simulations. For getting rid of such a problem, we applied the MTS SGS model developed by Inagaki et al. (2002). This model takes into account of the damping effect automatically by using a time scale calculated by the harmonic averaging of the SGS time-scale component and the time scale given by the strain rate. In the MTS SGS model, the eddy viscosity can be calculated by the following equations:

$$\nu_t = C_{MTS} k_{es} T_s \tag{4}$$

where

$$k_{es} = (\overline{u}_k - \hat{\overline{u}}_k)^2 \tag{5}$$

$$T_s^{-1} = \left(\frac{\overline{\Delta}}{\sqrt{k_{es}}} \right)^{-1} + \left(\frac{C_T}{|\overline{S}|} \right)^{-1} \tag{6}$$

C_{MTS} and C_T are the model parameters and take the value of 0.05 and 10, respectively, $\overline{\Delta}$ is the filtering scale defined by $\overline{\Delta} = (\Delta x \Delta y \Delta z)^{1/3}$, $|\overline{S}|$ is the absolute level of grid-scale strain rate, and $\hat{\overline{u}}_k$ is the filtered grid-scale velocity components. As mentioned previously, in addition to the above implementation, a density function method was introduced to the model to represent water surface variations. Finally, periodic boundary conditions were applied in longitudinal and transverse directions.

SIMULATION CONDITION

Two types of roughness features were considered in the present study, i.e. natural gravel beds with a diameter of 1.0 or 1.5cm. A DED with randomly distributed particles was measured by a stereoscopic method, in which images of the gravel particles placed on a plate were taken from two different angles normal to the plate and used to reconstruct the three dimensional gravel bed profile by a photogrammetric technique. The bottom profile obtained by the present method is shown in Fig.1. Although the individual gravel shape is not completely reconstructed, the random distribution of bottom roughness can be generated fairly well. The number of grid in LES was 85, 100 and 100 in streamwise, transverse and vertical directions, respectively for the gravel bed flow simulation. The number of vertical grid was relatively larger than conventional simulations because air phase calculation was required in the present LES model. The LES was conducted for the Froude numbers of 0.15, 0.3, 0.4 and 0.6 with a common depth of five centimeters. The range of the Reynolds number was between 10,500 and 21,000. The water depth was kept almost constant at about 5cm while changing the Froude number in the range of subcritical flow conditions.

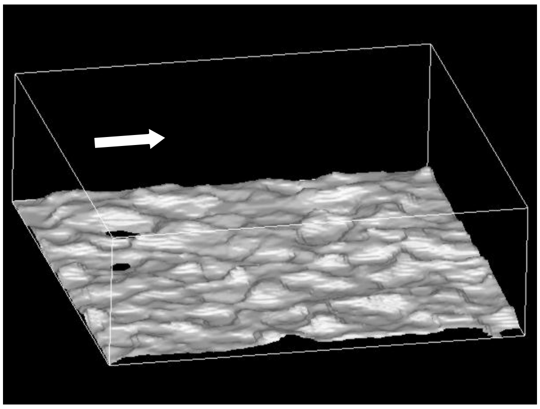
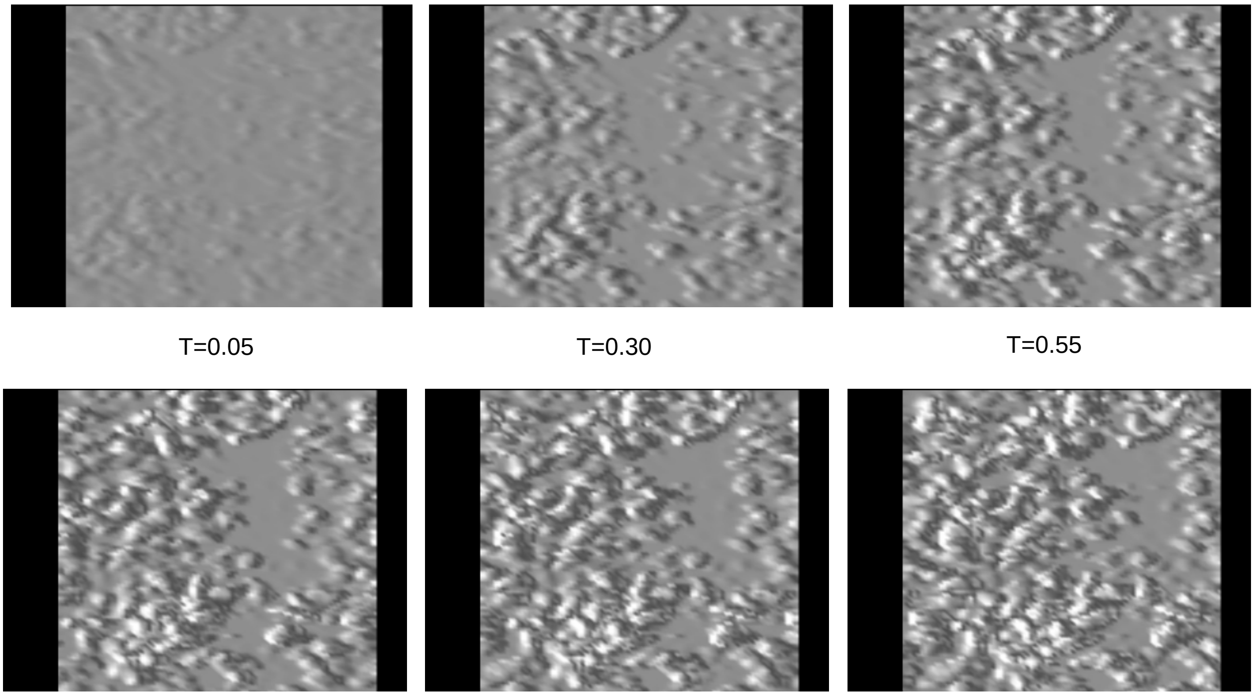


Fig.1 DED for gravel bed; 27.1cm in flow direction and 26.5cm in lateral direction

VISUALIZATION WITH PSEUDO-TRACER CLOUD

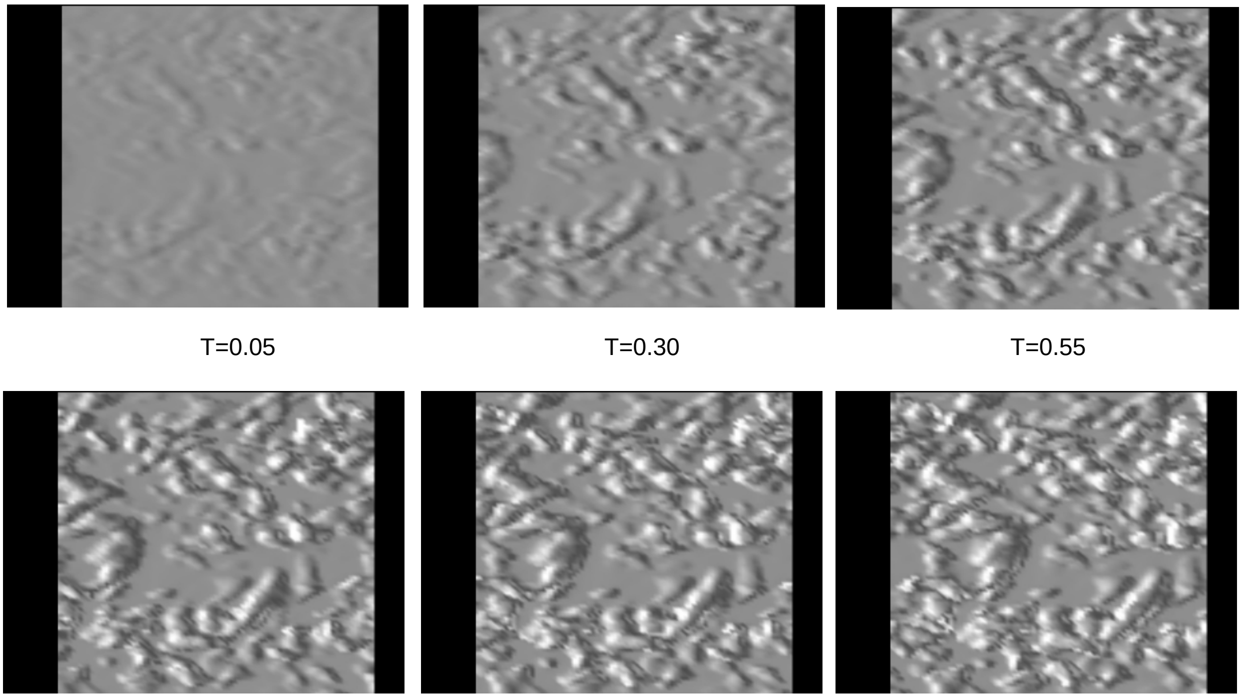
As previously mentioned, water surface fluctuation in turbulent flow might be caused by the impingement of vortices against the water surface. For investigating such a phenomenon, visualization of large-scale turbulence in a horizontal field near the water surface can be useful. In the light of this simple idea, we devised a method to use pseudo-particle tracers repeatedly placed in a horizontal plane. In this flow visualization, a large number of pseudo-tracers are firstly generated in a horizontal plane. After the particles are advected with their local velocity vectors, other pseudo-tracers are generated at the same horizontal location. While repeating this procedure at an appropriate frequency, three dimensional coordinates of every particle generated at any instance are stored on memory. The specific feature of the method is that the maximum height of the pseudo-particle location at each horizontal coordinates is preserved during the process in order to reveal the effects of upwelling vortices. Therefore, when we view the cloud of particles from above the horizontal plane, some large-scale flow structures become visible as a sort of topographic surface. This visualization method is similar to a hydrogen-bubble technique to a certain extent, in which bubbles are generated in a horizontal membrane. However, it is difficult to conduct such an experiment in a physical situation. In the present simulation, 200 by 200 pseudo-particles were traced at every non-dimensional time step of 0.05. As a typical example, the evolution of particle membrane located at y=0.8H for the case of the Froude number 0.6 and gravel diameter of 1.5cm is shown in Fig.2. Here, H is the water depth equal to the height of the liquid phase. The flow direction is from left to right. At T=0.05, many bumps with low height appear randomly in the entire membrane due to the upwelling fluid motions. Some of these bumps grow in height with time, but a downstream central portion is kept flat because the flow is ascending in this region. It can be noted that the flat portion gradually advected in the downstream direction while shrinking its size. Another feature of the visualized image is that the shape of the respective bump looks like an oval with longer diameter in the streamwise direction.



T=0.05 T=0.30 T=0.55

T=0.80 T=1.05 T=1.30

Fig.2 Evolution of membrane composed of pseudo-pareticles; at *y*=0.8*H* with Fr=0.6 for gravel diameter of 1.5cm, flow direction is from left to right.



T=0.05 T=0.30 T=0.55

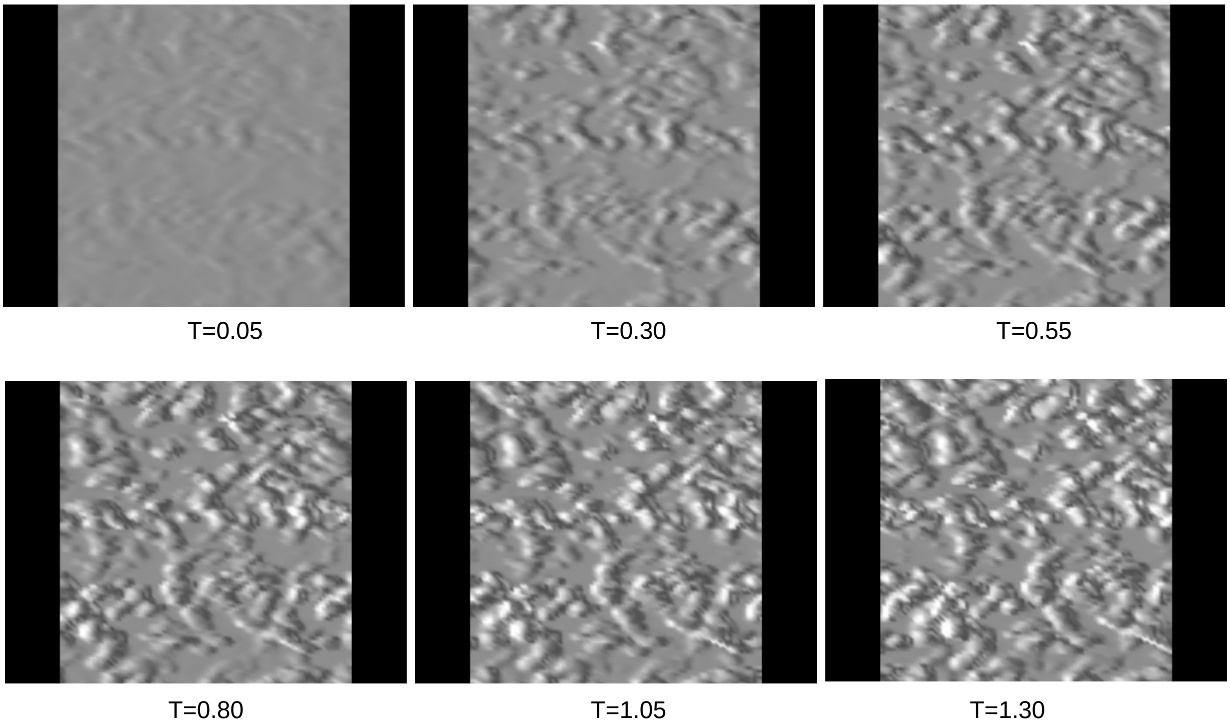
T=0.80 T=1.05 T=1.30

Fig.3 Evolution of membrane composed of pseudo-pareticles; at *y*=0.8*H* with Fr=0.15 for gravel diameter of 1.5cm, flow direction is from left to right.

DISCUSSIONS

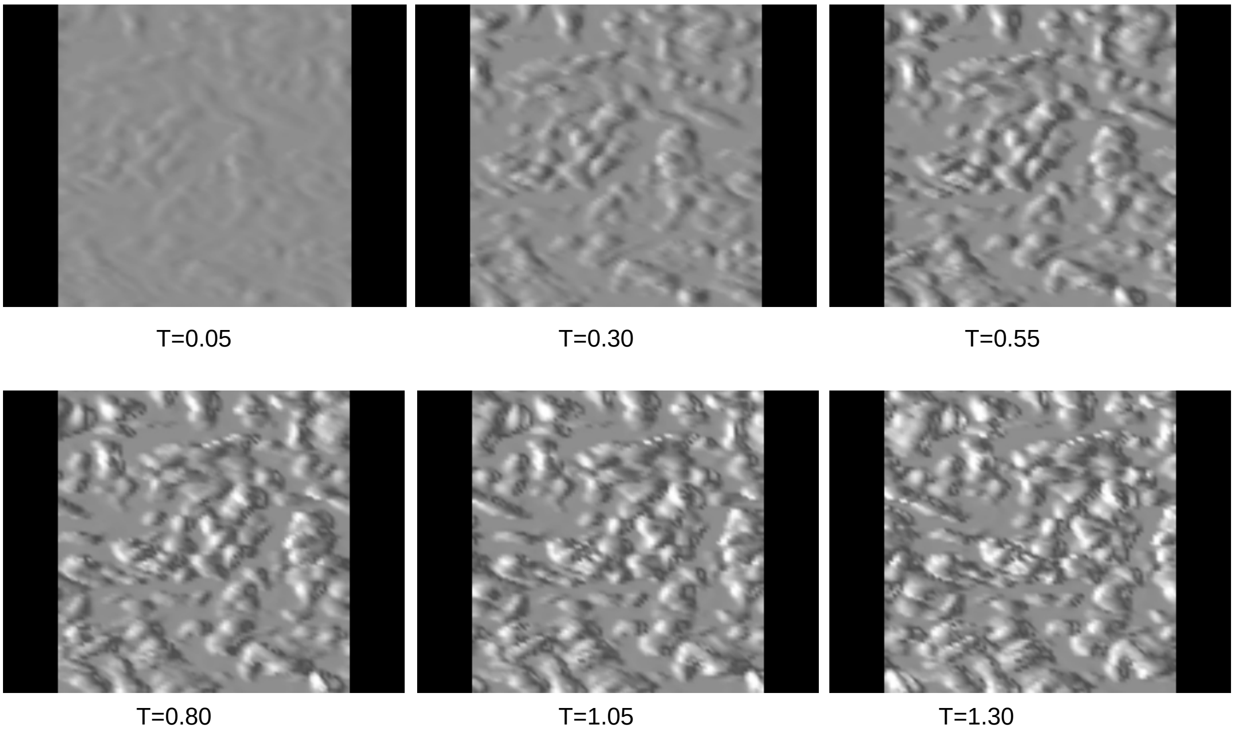
In order to compare the effects of the Froude number, a similar expression to Fig.2 for the lower Froude number of 0.15 with the same diameter is shown in Fig.3. It is interesting to note that the scale of each bump is slightly larger than the case of Fr=0.6, with a shape elongated or skewed in the downstream direction. In addition, the flat area is relatively larger than Fr=0.6, suggesting ascending flows are generated in between the upwelling flow. Another aspect in Fig.3 is that the bumps are sparsely distributed. These features may be the cause of less fluctuation for a lower Froude number.

The results for a rough bed with a smaller gravel diameter of 1.0cm are shown in Fig.4 and Fig.5. Generally speaking, the scale and the distribution density became smaller compared with the previous cases with a larger gravel diameter. In Fig.4 for Fr=0.6, the shape of each bump is round but its size is smaller than that indicated in Fig.2 for the same Froude number. Furthermore, the height of each bump is still low even at T=0.55, which suggests that the upwelling velocity of each vortex is relatively smaller than the previous cases. In Fig.5 for the case of Fr=0.15, general aspects are similar to Fig.4, except that the size of the bump is larger than the case for Fr=0.6. This feature was observed for the cases of larger gravels. The interesting pattern observable in Fig.5 is that a pair of long and thin bumps with its axis slightly tilted in transverse direction is generated in between larger bumps. This suggests that a pair of vortex tube with different rotational directions was produced by the ambient structure of vortex field. Another feature commonly observed in Fig.4 and Fig.5 is the distribution density of the bumps is uniform with less local accumulation of vortex structures as seen in Fig.2 and Fig.3. Finally, it can be commonly observed that the structure visualized in a form of the bumps is advected in the downstream direction at almost uniform local speed, except that the advection speed of a larger scale structure tends to be decelerated in the streaming process.


T=0.05 T=0.30 T=0.55

T=0.80 T=1.05 T=1.30

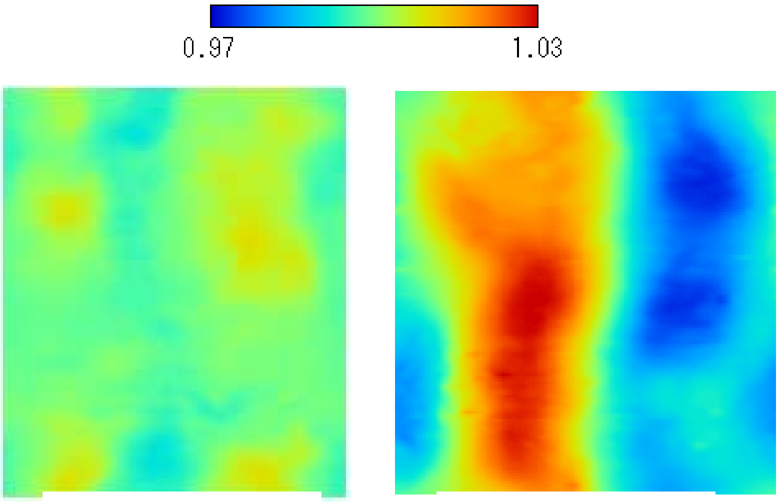
Fig.4 Evolution of membrane composed of pseudo-pareticles; at *y*=0.8*H* with Fr=0.6 for gravel diameter of 1.0cm, flow direction is from left to right.



T=0.05 T=0.30 T=0.55

T=0.80 T=1.05 T=1.30

Fig.5 Evolution of membrane composed of pseudo-pareticles; at y=0.8H with Fr=0.15 for gravel diameter of 1.0cm, flow direction is from left to right.



(a) d=1.0cm (a) d=1.5cm

Fig.6 Comparison of water surface profile for Fr=0.6, flow direction is from left to right.

In order to examine the effect of the flow structure just beneath the water surface, water surface profiles for different bed roughness sizes at T=0.30 are compared in Fig.6 for the case of Fr=0.6. Please note that the water surface variations are shown with the same depth range between 0.97H and 1.03H. It is obvious that the water surface shape is different for a different bed roughness. For a gravel diameter of 1.0cm, the water surface is almost flat with a little variation of about one-percent change. The areas with higher or lower water level are distributed in a patch-like pattern but its pattern does not seem to have a direct relationship to the underwater structure shown in Fig.4. On the other hand, for the case of d=1.5cm, the water surface shows much larger variation with more than three-percent change. At the same time, the water surface shows an undulation that corresponds to the underwater structure just beneath the water surface, i.e. larger upwelling flows are generated in the upstream portion of the region while ascending flows can be observed in the downstream region. For the cases with lower Froude numbers, the water surface profile didn't show appreciable variations, probably because the strength of the upwelling flow is not strong enough to deform the water surface itself.

CONCLUSIONS

By using an LES model capable to treat water surface variations and arbitrary bottom topography with roughness elements, the characteristics of turbulent flow structures just beneath the water surface were examined by devising a novel flow visualization method. In this method, large-scale turbulent structures generated by upwelling vortices are displayed in a form of topographic geometry, in which vortices passing through a horizontal plane are expressed as a group of bumps. It was made clear that the present visualization method can demonstrate the difference of turbulent structures appeared in a horizontal plane. The commonly observed feature is that the size of the bump becomes relatively smaller for a larger Froude number as the Reynolds number also increases for the same condition. The change of the scale should be related to the change of the integral scale of the flow with hydraulic parameters such as the Froude number or the Reynolds number; however, the visualization of the independent vortex distribution has been rarely performed in the past researches. Another interesting feature made clear in the present simulation was that the scale of each bump visualized in the present method seems comparable to the size of the bottom roughness, which suggests that a lump of fluid separated at each roughness element has reached near the water surface by preserving its relative scale because the water depth is relatively shallow in the present simulation. However, further research is required to understand the effects of roughness size on a turbulent scale and the interrelationship between roughness and water surface variation and fluctuation. The experimental investigation is also needed to evaluate the findings estimated in the numerical simulation.

REFERENCES

Bates, P.D., Lane, S. N. And Ferguson, R. I. (2005) Computational Fluid Dynamics, Applications in Environmental Hydraulics, John Wiley &Sons.

Fadln,E. A., Verzicco, R., Orlandi, P. and Mohd-Yusof, J. (2000) Combined Immersed- Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations, *Journal of Computational Physics*,161-1, 35-60.

Fujita, I., Watanabe, H. and Tsubaki, R. (2007) Development of a non-intrusive and efficient flow monitoring technique: The space time image velocimetry (STIV), International Journal of River Basin Management, Vol.5, No.2, pp.105-114.

Germano, M., Piomelli, U., Moin, P. and Cabot, W. H. (1991) A dynamic subgrid-scale eddy viscosity model, *Phys. Fluids*, A3, 7, 1760-1765.

Grass A. J. (1971) Structural features of turbulent flow over smooth and rough boundaries, Journal of Fluid Mechanics, 50, 233-255.

Fujita, I., Kadotani, K., Matsubara, T. and Tsubaki, R. (2008) Analysis of water surface oscillation at open-channel side cavity by image analysys and large eddy simulation, Advances in Hydro-Science and Engineering, Volume VIII, on CDROM.

Graf and Altinakar (1998) Fluvial hydraulics, Wiley, New York.

Cui, J., Patel, V. C. and Lin, C. L. (2003) Prediction of turbulenct flow over rough surfaces using a force field in large eddy simulation, Transactions of the ASME, 125, 2-9.

Inagaki, M., Kondo, T. and Nagano, Y. (2002) A mixed-time-scale SGS model with fixed model-parameters for practical LES, Engineering Turbulence Modeling and Experiments 5 (Edited by Rodi, W and Fueyo, N.), Elsevier, 257-266.

Jimenez, J. (2004) Turbulent flows over rough walls, Annual Review of Fluid Mechanics, 36, 173-196.

Naot, D., Nezu, I. and Nakagawa, H. (1996) Hydrodynamic behaviour of partly vegetated open channels, Journal of Hydraulic Engineering, 122, 625-633.

Nezu, I, and Nakagawa, H. (1993) Turbulence in open-channel flows, Balkema, Rotterdam, The Netherlands.

Nikora, V, Goring, D., McEwan, I. and Griffiths, G. (2001) Spatially averaged open-channel flow over rough bed, Journal of Hydraulic Engineering, 127.2, 123-133.

Xiaohui, S. and Li, C. W. (2002) Large eddy simulation of free surface turbulent flow in partly vegetated open channels, International Journal for Numerical Methods in Fluids, 39, 919-937.

Muste, M., Fujita, I., and Hauet, A. (2008) Large-scale particle imagevelocimetry for measurements in riverine environments, Water Resources Research, 44, W00D19, doi:10.1029/2008WR006950.