ANALYSIS OF OPEN-CHANNEL FLOW WITH STRIP ROUGHNESS BY LES USING IMMERSED BOUNDARY METHOD

HIDETO YOSHIMURA[1]

[1]Graduate School of Science and Technology, Kobe University, 1-1 Rokkodai, Nada, Kobe, 657-8501, Japan
phone/fax: +81-78-803-6439, hideto_y0312@yahoo.co.jp

ABSTRACT

Open channel flow with strip roughness is investigated by a large eddy simulation (LES) using an immersed boundary method to represent the arbitrary bottom roughness shape. In the simulation a sophisticated SGS model, the MTS-SGS model proposed by Inagaki et al. (2002), is used to simulate the flow near the boundary without a damping function. In addition to the simulation, flow visualization experiments using the particle image velocimetry (PIV) are performed to compare the results with simulation. The hydraulic conditions for the simulation and experiment are restricted to a relatively small Froude number (Fr) flow, e.g. Fr less than 0.25, in order not to generate appreciable water surface fluctuation in the present research. After confirming that distribution of statistical properties such as turbulent intensities calculated by LES agrees fairly well with those by PIV, we paid attention to the generation of vortices influencing the water surface velocity field. It was made clear that the proposed LES model is capable to simulate the time-dependent features of the vortices separated at the strip roughness and the three-dimensional structure of the separation vortex just reaching the water surface.

Keywords: *LES, strip roughness, open-channel turbulence, PIV, immersed boundary method, separation vortex*

1 INTRODUCTION

It is well known that river surface configuration changes significantly depending on the physical effects such as wind, resistance by river structures or turbulence itself. What can be observed usually in the actual river surface is caused by the combination of the above effects. From a microscopic point of view, the flow near the water surface is important because there occurs an active gas exchange at the interface between air and liquid phases (Jahne and Hauβecker 1998). From a macroscopic viewpoint, since it can be considered that the convection speed of water surface configuration during flood is closely related to the water surface velocity, this phenomenon can be used to estimate river surface velocity distributions and further the river flow discharge with the information of water depth distribution (Fujita et al. 1998, Fujita and Tsubaki 2002). On the other hand, the analysis techniques for three-dimensional turbulent flows have developed significantly in the past decades; in addition to the Reynolds averaged numerical simulation (RANS) models, the direct numerical simulation (DNS) models or the large eddy simulation (LES) models have come to be used in the practical applications. However, most of the simulation models and their applications have dealt with flows with smooth boundary conditions and there are few simulation examples treating channel flows with roughness (Ikeda and Durbin 2002). Simulation models that can handle open-channel flows with free-surface variations are also available in recent years (Scardovelli and Zaleski 1999, Shen and Yue 2002); however, fluctuation of open-channel water surface fields arising from rough wall turbulence has not been discussed in detail numerically and experimentally. Therefore, in the present research, we pay attention to the generation of separation vortices by roughness elements and their effects on the water surface flow field. In order to investigate such flow feature, we developed a large eddy simulation model capable to treat arbitrary boundary shape. We also performed flow visualization experiments and the detailed flow features obtained the particle image velocimetry (PIV) are compared with the simulated results. In the present research, for the purpose of examining fundamental flow characteristics, open-channel flows with rectangular strip roughness under relatively small Froude number conditions are investigated.

2 OUTLINE OF EXPERIMENTS

The experiments are conducted using a straight acrylic open-channel flume with a length of 4m and a width of 0.2m. As a roughness element, we used an aluminum square bar with a side length of 0.9 cm and a length of 0.2m. The longitudinal vertical cross-section of the channel is visualized by a laser light sheet and the visualized flow seeded by nylon power with an average diameter of 10μm is recorded using a high speed CCD camera of 110fps.

The hydraulic condition in the experiments is indicated in table 1. The water depth is kept constant at 4.0cm and the Froude number is set at 0.25 so that water surface becomes almost stable in the entire flow region. The Froude number just above the roughness element becomes 0.37. We performed the experiment for the two relative roughness spacing, L/k, five and ten, where L is the spacing between the center of the roughness element and k is the height of the element.

Table 1 Hydraulic condition

	Ls05	Ls10
water depth：h_1(cm)	4.0	
mean velocity：U_m(cm/s)	15.6	
relative roughness spacing：L/k	5	10
Froude number：Fr	0.25	
Reynolds number：Re	6240	
channel slope：I	1/500	

3 NUMERICAL SIMULATION MODEL

3.1 BASIC EQUATIONS

The numerical simulation is based on the LES in the Cartesian grid system. The basic 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 is to be noted that the unsteady term in the filtered Navier-Stokes equation is discretized by a forward differencing.

3.2 IMMERSED BOUNDARY METHOD

Actual river beds have rough surface composed of complicated arrangements of sand particles or gravels and they will be deformed significantly when sand waves are generated. In order to represent such bed shape in numerical simulations, it is more appropriate to use a method capable to define arbitrary bed shape relatively easily in the Cartesian coordinate system. Therefore, 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 introducing an external force into the basic 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* is the summation from the first to the fourth terms in the right hand side of the equation(2). The term U_i^{n+1} is the velocity vector interpolated by using the value at the near-wall grid and the value at the boundary normally chosen as zero. The term becomes zero within the wall grids.

3.3 MTS SGS MODEL

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 becomes difficult to measure such a distance for a complicated topography. The dynamic SGS model developed by Germano et

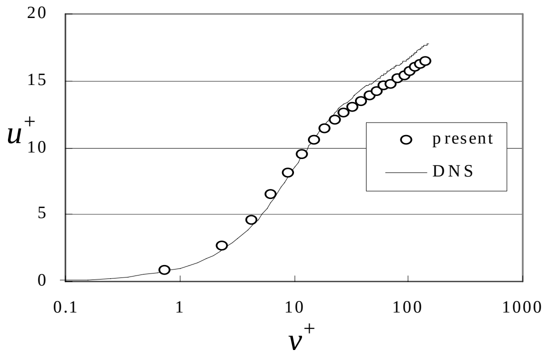


Fig.1 Mean velocity distributions

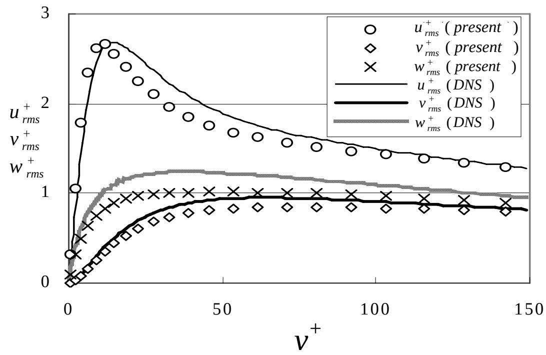


Fig.2 Turbulent intensity distributions

al.(1991) is capable to solve this problem, however this model tends to become unstable in the simulation. Hence, 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:

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

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

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

where C_{MTS} and C_T are the model parameters and take the value of 0.05 and 10, respectively, $\bar{\Delta}$ is the filtering scale defined by $\bar{\Delta} = (\Delta x \Delta y \Delta z)^{1/3}$, $|\bar{S}|$ is the absolute level of grid-scale strain rate, and $\hat{\bar{u}}_k$ is the filtered grid-scale velocity components.

3.4 SIMULATION CONDITION

In the present analysis, the simulation domain is set at 9*H* by *H* by *H*, where *H* is the mean water depth. The domain is discretized into 180 by 48 by 48 grids with a uniform grid spacing in spanwise direction and non-uniform grid spacing in the other directions. A cyclic boundary condition is applied in the streamwise and the spanwise directions, the slip condition at the water surface and the non-slip condition at the bottom boundary. We use the fractional step method for solving the system equations with the second order Adams-Bashforth scheme for the unsteady term and the Crank-Nicolson scheme for the molecular viscosity terms. The non-dimensional time step used is set at 5.0×10^{-4}.

4 RESULTS AND DISCUSSIONS

4.1 UNIFORM CHANNEL FLOW SIMULATION

The developed LES model is first applied to the uniform channel flow as a preliminary examination. Here, the flat bottom boundary is expressed by the IB method. The turbulent flow for the frictional Reynolds number of 300 is calculated with a 48x48x48 grid system. The calculated streamwise mean velocity distribution is compared with the DNS result as shown in Fig.1. As shown in the figure the overall agreement with the DNS result is favorable except near the upper region. The comparison of turbulent intensities is indicated in Fig.2. The general profiles agree well with those obtained by DNS.

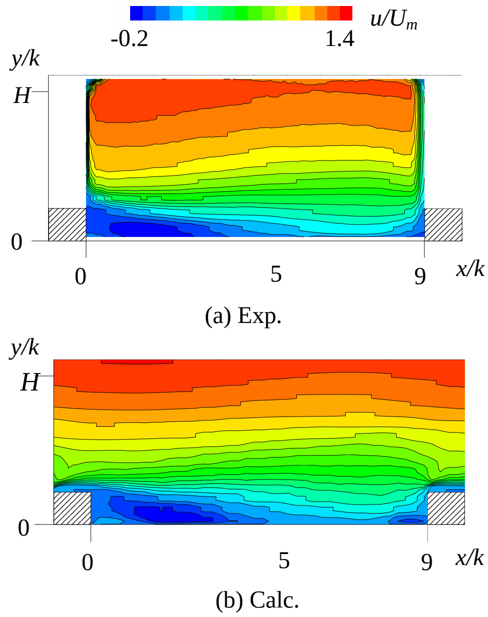


Fig.3 Mean velocity distribution (Ls10)

4.2 STRIP ROUGHNESS FLOW SIMULATION

The mean velocity distributions obtained by the experiment and the simulation are compared in Fig.3. The calculated profile agrees well with that by the experiment including the reattachment length between the strip roughnesses. The vertical turbulent intensities for the cases of Ls05 and Ls10 are indicated in Fig.4. It can be seen that the stochastic turbulent properties can be represented fairly well by the developed LES model.

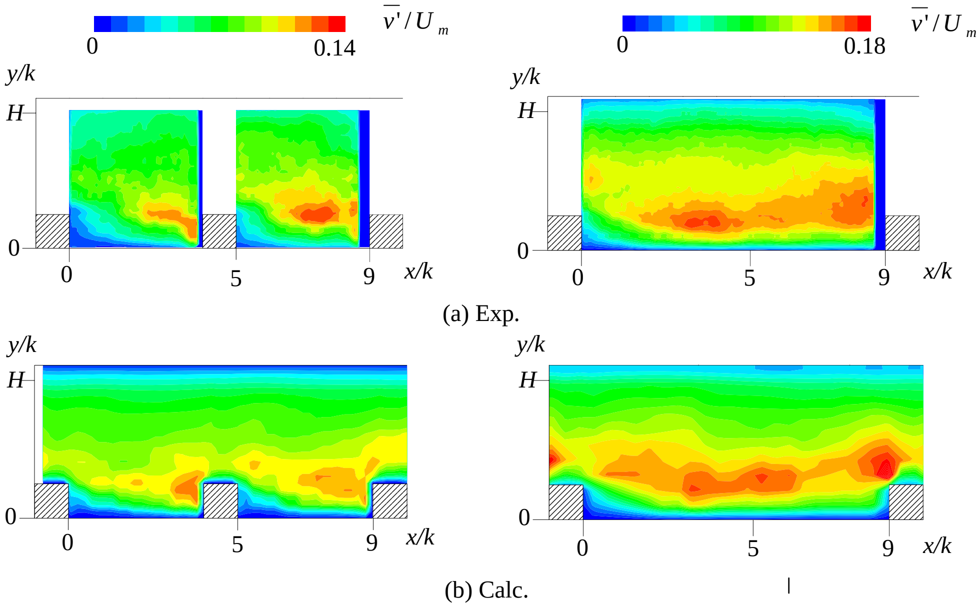


Fig.4 Turbulent vertical intensity distributions (left:Ls05, right：Ls10)

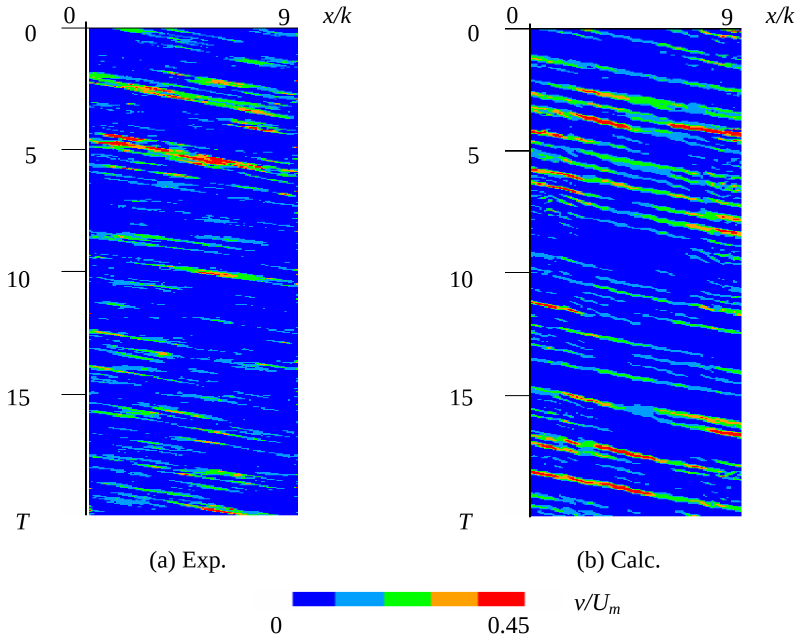
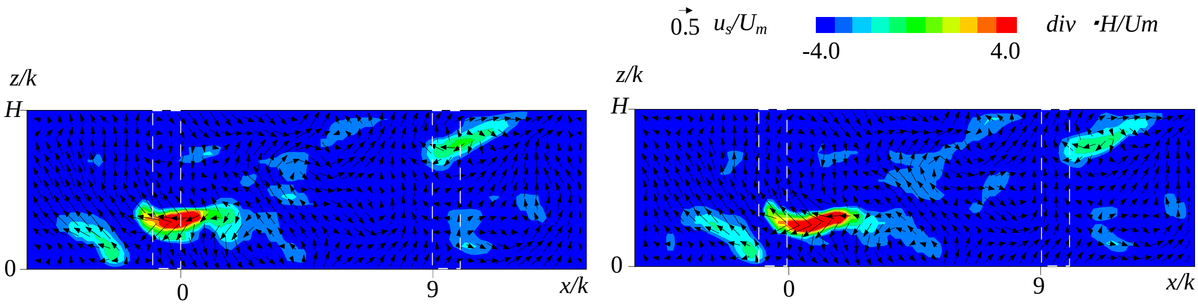


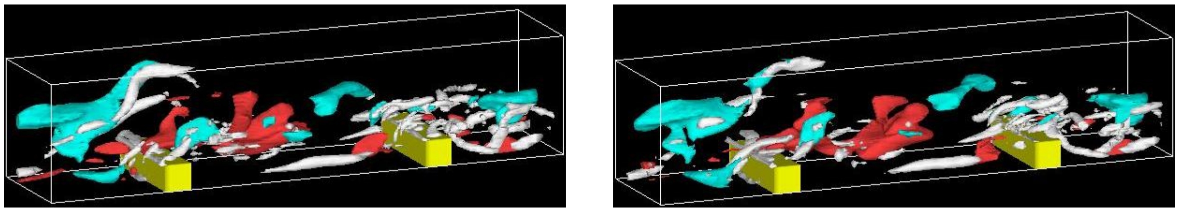
Fig.5 Space time expression of vertical velocity component near the water surface for LS10 ($T=tU_m/h_1$)

It was observed in the preliminary experiment that boil vortices are generated actively at the water surface between the roughness elements. This can be due to the ascending vortices separated at the strip roughness. In order to examine the time-dependent feature of the flow, the vertical velocity distributions close to the water surface are compared. Fig.5 shows the space-time expression of the vertical velocity distribution along the longitudinal line segment located horizontally near the water surface for the case of LS10. The time evolution of the velocity distribution at the height of y/h_1=0.9 is obtained from the PIV results and that at y/H=0.98 is obtained from the LES results. To make clear the intermittent feature of the flows, only the positive components are displayed in Fig.5. It is clearly seen that LES yields quite a similar intermittent flow feature to that obtained by PIV; i.e. local flows with relatively larger vertical component are generated at some time spacing and they are convected in the downstream direction almost at a constant speed. It should be noted that stronger vertical flows have larger lifetimes and they tend to appear in a clustered form.

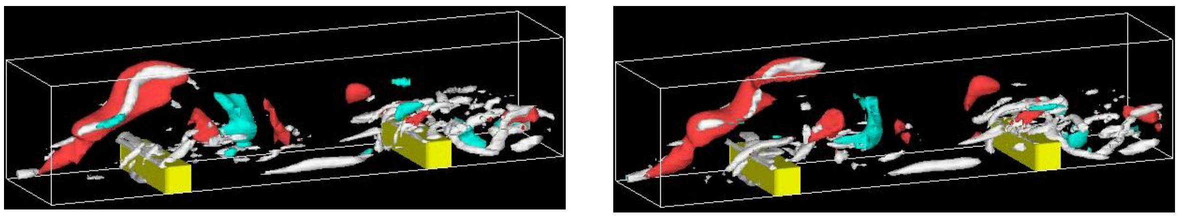
The three-dimensional structure of the flow and its effect on the water surface flow field are investigated by the LES model. Fig.6(a) shows the surface divergence distribution together with the instantaneous surface velocity vectors observed from a frame moving with the mean surface velocity. It can be seen that the velocity field is decelerated when a large positive divergence region appears locally near the water surface. In order to examine the flow structure just beneath the water surface for the same instances, three-dimensional distributions of vortex core, high and low streamwise velocity region and vertical velocity region are displayed in Fig.6(b) and Fig.6(c). Here, the vortex core region is displayed as an



(a) Distribution of positive divergence and velocity vectors observed from a frame moving with the mean velocity at the water surface



(b) Distributions of vortex core(white) and high(red) and low(blue) streamwise velocity component



(c) Distributions of vortex core(white) and upward(red) and downward(blue) vertical velocity component

$T = 0$ $\qquad\qquad$ $T = 0.25$

Fig.6 Three-dimensional flow features and their effect on water surface flow field (T=tU_m/h_1); vortex core region is defined by the second invariant Q=10

equi-valued surface of the second invariant Q=10 and visualized as white surface. It is seen that a large vortex-tube developed just upstream of the strip roughness has reached the water surface and subsequently the vortex-tube is elongated in the downstream direction due to the strong shear stress in the main flow. It should be noted from Fig.6(b) that such vortex cores are associated with lower velocity and from Fig.6(c) that near-surface vortex cores are related to the region of upward velocity. This result suggests that the vortices with lower momentum generated at the bottom ascend up to the water surface, which can explain the features of the surface velocity vector field indicated in Fig.6(a).

5 CONCLUSIONS

Open-channel flows with simplified roughness elements are investigated by a flow visualization experiment and a numerical simulation in order to examine the fundamental effects of separated vortices on the water surface velocity field. As a result, it was made clear that the developed LES model is capable to simulate the time-dependent vortex flow feature as well as the statistical turbulent properties. Since the hydraulic condition in the present research is confined to relatively low Froude number, further development of the simulation model is necessary which can take into account of the large deformation of water surface with complicated bottom roughness arrangements.

ACKNOWLEDGEMENT

The author is grateful to the support by Prof. Ichiro Fujita at the department of civil engineering, Kobe University.

REFERENCES

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, pp.35-60.

Fujita, I., Muste, M. and Kruger, A. (1998), Large-scale particle image velocimetry for flow analysis in hydraulic engineering applications, *Journal of Hydraulic Research*, Vol.36, No.3, pp.397-414.

Fujita, I. and Tsubaki, R. (2002), A novel free-surface velocity measurement method using spatiotemporal images, *Hydraulic Measurements and Experimental Methods*, *ASCE*, on CDROM.

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

Ikeda, T. and Durbin, P.A. (2002), Direct simulations of a rough-wall channel flow, Report No.TF-81, *Flow Physics and Combustion Division*, Stanford University.

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, pp.257-266.

Jahne, B. and Hauβecker, H.(1998), Air water gas exchange, *Ann. Rev. Fluid Mech.*, Vol.30, pp.443-468.

Scardovelli, R. and Zaleski, S. (1999), Direct numerical simulation of free-surface and interfacial flow, *Ann. Rev. Fluid Mech.*, Vol.31, pp.567-603.

Shen, L. and Yue, D. K. P. (2001), Large-eddy simulation of free-surface turbulence, *J. Fluid Mech.* ,Vol.440, pp.75-116.