

STUDY ON FLOOD FLOW CONSIDERING WATER LEVEL RISE BY HYDRAULIC STRUCTURES

Dongkeun LEE¹, Hajime NAKAGAWA², Kenji KAWAIKE³, Yasuyuki BABA⁴ and Hao ZHANG⁴

¹Student Member of JSCE, Graduate Student, Department of Civil and Earth Resources Engineering, Kyoto University (Katsura Campus, Nishikyo-ku, Kyoto 615-8540, Japan)

²Member of JSCE, Dr. of Eng., Professor, Disaster Prevention Research Institute, Kyoto University (Shimomisu, Yoko-oji, Fushimi-ku, Kyoto 612-8235, Japan)

³Member of JSCE, Dr. of Eng., Associate Professor, Disaster Prevention Research Institute, Kyoto University (Shimomisu, Yoko-oji, Fushimi-ku, Kyoto 612-8235, Japan)

⁴Member of JSCE, Dr. of Eng., Assistant Professor, Disaster Prevention Research Institute, Kyoto University (Shimomisu, Yoko-oji, Fushimi-ku, Kyoto 612-8235, Japan)

A 3D numerical model for the estimation of flow considering water level rise by hydraulic structures is presented and the results are compared with the experimental results. The numerical simulation is conducted on the unstructured meshes with finite volume method. The volume of fluid method is used to represent the free surface and the standard $k-\varepsilon$ model is employed to calculate the turbulent flow. In this study, the numerical simulation and the laboratory experiment are performed about the bridge which is the representative hydraulic structure. The validity of the developed numerical model is considered through the obtained data from and the effect of water level rise by hydraulic structure is estimated. From the computational results, it is found that the model is able to reproduce the flood flow considering water level rise by hydraulic structures with a reasonable accuracy. Therefore, the numerical model developed in this study will be useful to estimate the water level rise and overflow discharge caused by hydraulic structures.

Key Words : 3D numerical model, unstructured mesh, volume of fluid method, water level rise, backwater, hydraulic structures

1. INTRODUCTION

Hydraulic structures such as pier and bridge girder cause water level rise during flood situations, which is of great interests in engineering practices. In this study, a numerical model is proposed to simulate the flow considering water level rise by hydraulic structures. The simulation is related to free surface flow in three dimensional computations. In numerical simulations of open channel flow, free surface is usually replaced by a rigid lid. This approach is suitable only if free surface is non-complex. For rapidly changing free surface, this approximation will introduce nonphysical errors. There are many capturing methods available to

simulate the free surface. One of the most successful methods has been the volume of fluid method proposed by Hirt and Nichols¹⁾. This method's popularity is based on ease of implementation, accuracy and computational efficiency. The method has been used by several researchers to capture the free surface on structured mesh. The method is a powerful approach, but it is not known to have been implemented on unstructured meshes. Also, instances of non-physical deformation of the interface shape have been reported^{2), 3), 4)}. In this paper, a high resolution scheme proposed by Ubbink and Issa⁵⁾ is used to capture the free surface. This scheme treats the volume fraction of each fluid which is used as the weighting factor to get the fluid properties. Particularly the method of Ubbink and

Issa⁵⁾ is adaptive to arbitrary unstructured meshes. The computation is implemented into a finite volume procedure using unstructured meshes. The proposed methodology is applied to the flow including hydraulic structures.

2. GOVERNING EQUATIONS

(1) Fluid flow model

The governing equations for continuity and momentum with the tensor notation are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = g_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

where t is the time, u_i is the time-averaged velocity, x_i is the Cartesian coordinate component, p is the time-averaged pressure, ν is molecular kinematic viscosity, $\tau_{ij} = -\overline{\rho u_i u_j}$ are the Reynolds stress tensors, u_i is the fluctuating velocity component.

The conservative form of the scalar convection equation for the volume fraction α is:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial \alpha u_i}{\partial x_i} = 0 \quad (3)$$

(2) Turbulence model

In the standard $k - \varepsilon$ model, the Reynolds tensors are acquired through the linear constitutive equation:

$$-\overline{u_i u_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (4)$$

where k is the turbulent kinetic energy, δ_{ij} is the Kronecker delta, ν_t is the eddy viscosity and S_{ij} is the strain rate tensor defined as:

$$\delta_{ij} = \begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j \end{cases} \quad (5)$$

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (6)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (7)$$

where ε is the turbulent energy dissipation rate.

Two transport equations are employed to estimate k and ε :

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G - \varepsilon \quad (8)$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + (C_{1\varepsilon} G - C_{2\varepsilon} \varepsilon) \frac{\varepsilon}{k} \quad (9)$$

where G is the production rate of the turbulent kinetic energy k and is defined as:

$$G = -\overline{u_i u_j} \frac{\partial u_i}{\partial x_j} \quad (10)$$

The constants in equations (6), (8) and (9) take the values suggested by Rodi⁶⁾ and generally the universal values are as follows:

$$C_\mu = 0.09 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92 \quad \sigma_k = 1.00 \quad \sigma_\varepsilon = 1.30 \quad (11)$$

3. NUMERICAL METHODS

(1) Discretization methods

The finite volume method based on the unstructured mesh is employed in the model. The governing equations are integrated over a number of polyhedral control volumes covering the whole domain in the finite volume method, the general form is as follows:

$$\frac{\partial}{\partial t} \int_V \phi dV + \int_S \phi u \cdot n dS = \int_S \Gamma \nabla \phi \cdot n dS + \int_V b dV \quad (12)$$

where V is the volume of the control volume, S is the surface of the control volume with a unit normal vector n directing outwards, ϕ is the general conserved quantity representing either scalars or vector and tensor field components, Γ is the diffusion coefficient and b is the volumetric source of the quantity ϕ .

The equation system is mesh independent and is valid for arbitrary polyhedral control-volumes. The conserved equations are discretized on a collocated unstructured mesh. The surface fluxes are calculated from the Rhie-Chow interpolation⁷⁾ to avoid the checkerboard effect. The second order implicit Crank-Nicolson scheme is employed in the temporal integral. The widely used SIMPLE(Semi-implicit method for pressure-linked equations) method is used for the coupling of the pressure and the velocity.

(2) Solution methods

The final algebraic equation systems resulted from the discretization process are characterized by sparse and non-symmetrical coefficient matrices. They are solved with a preconditioned GMRES(generalized minimal residual method) incorporated with an ILUTP(incomplete LU factorization with threshold and pivoting) preconditioner proposed by Saad⁸⁾. The relaxation method proposed by Patankar⁹⁾ is implemented to

increase the diagonal dominance of the coefficient matrices.

(3) Volume fraction convection equation

The finite volume discretization of the volume fraction convection equation is based on the integral form of equation (3) over each control volume and the time interval. If P denotes the centre of the control volume, the Crank-Nicolson discretization, second-order accurate in time, leads to:

$$(\alpha_P^{t+\Delta t} - \alpha_P^t)V_P = -\sum_{f=1}^n \frac{1}{2} ((\alpha_f F_f)^t + (\alpha_f F_f)^{t+\Delta t}) \Delta t \quad (13)$$

where f is the centroid of the cell face and F_f is the volumetric flux defined as:

$$F_f = S_f \cdot u_f \quad (14)$$

where S is the face area with unit normal vector directing outwards. The summation in equation (13) is over all cell faces.

If the operator splitting is avoided and the solution is free from numerical diffusion in all flow directions, it should be noted that the Crank-Nicolson scheme is necessary. The scheme is more expensive in terms of computer storage because it needs both values of old and the new time level for the volumetric flux F at the faces. However, this can be overcome because for sufficiently a small time step the variation of F is negligible in comparison with the larger variation of α . Therefore, it is reasonable to use only the most recent value of F . The equation (13) reduces to:

$$\alpha_P^{t+\Delta t} = \alpha_P^t + \frac{\Delta t}{V_P} \sum_{f=1}^n \alpha_f^* F_f \quad (15)$$

where α_f^* is the approximation of the time-averaged volume fraction value at face.

The scheme proposed by Ubbink and Issa⁵⁾ is chosen to capture the free surface. The whole domain is treated as the mixture of water and air. The volume fraction is used as the weighting factor to get the mixture properties such as density and viscosity.

Most of the methods applied in volume fraction convection are employed the fractional steps or operator-splitting method on two or three dimensional problems. Ubbink and Issa⁵⁾ proposed CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes) method, in which through semi implicit disposal the volume fraction convection equation can be solved for two or three dimensions. Particularly, the method is efficient even if on unstructured meshes.

In this method, the concept of normalized variable diagram and the main idea of Hirt and

Nichols¹⁾ are employed to select diffusive scheme or dispersion scheme according to the direction of interface. And, a switch parameter between diffusive and dispersion scheme is introduced to improve the accuracy of diffusive and dispersion scheme.

In the above mentioned volume fraction convection equation, the volume fraction at a face can be written as:

$$\alpha_f^* = (1 - \beta_f) \frac{\alpha_D^t + \alpha_D^{t+\Delta t}}{2} + \beta_f \frac{\alpha_A^t + \alpha_A^{t+\Delta t}}{2} \quad (16)$$

where β_f is the weighting factor.

4. BOUNDARY CONDITIONS

The boundary conditions include the inlet, the outlet, the impermeable wall.

For the inlet boundary, it is generally considered as a Dirichlet boundary and all the quantities have to be prescribed. The turbulence quantities such as k and ε are also set as constant and Neumann boundary with zero gradients is applied to the pressure. At the outlet boundary, the outlet is set as far downstream of the study domain as possible and Neumann boundary with zero gradients can be assumed. The following correction method¹⁰⁾ is employed to ensure the conservation of mass.

$$u_{\hat{n}} = u_{\hat{n}}^{old} \frac{\sum_{inlet} u_{f\perp} S_f}{\sum_{outlet} u_{f\perp} S_f} \quad (17)$$

Near the impermeable wall, the flow velocity is assumed to be parallel to the wall. The standard wall function approach is used to link the turbulent domain with the near-wall area. The turbulence kinetic energy k and the dissipation rate ε are specified corresponding to a viscosity ratio of 10.0 and taking the turbulence intensity 8%. In this study, the total number of 24,890 polyhedral meshes is used. The unstructured mesh consists of hexahedra in order to represent the computational domain accurately, in particular the shape of the pier and girder.

5. LABORATORY EXPERIMENT

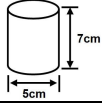
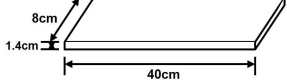
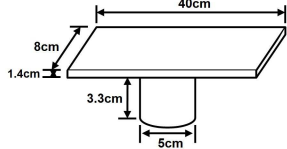
The objective of the laboratory experiment is to compare the variations of flow according to a kind of river structures under the same hydraulic conditions. And, the laboratory experiment is performed to compare with the numerical results.

The experimental channel used in this study is located at Ujigawa Open Laboratory, DPRI, Kyoto University and is straight channel of width 40cm, depth 23cm, and length 14.6m.

Table 1 Hydraulic conditions (uniform flow)

Parameters	Symbols(unit)	Values
Flow discharge	$Q(l/s)$	7.00
Water depth	$h_0(cm)$	4.76
Slope	I	1/987
Mean velocity	$u_m(cm/s)$	36.80
Reynolds number	Re	17,517
Froude number	Fr	0.54

Table 2 Experimental cases

	Structures	Figure
Case-1	No structures	
Case-2	Cylinder pier	
Case-3	Girder	
Case-4	Cylinder pier + Girder	

The detail of the experimental setup is described in Table 1. The experimental cases are shown in Table 2.

Case-1 is conducted to compare with the other cases. Case-2, Case-3 and Case-4 are the experiments to consider the effect of water level rise by the pier, girder and bridge, respectively.

In the experiment, the distributions of velocity are measured at the free surface and $z=2\text{cm}$ measured from the bottom. And, the water level is also measured for the shape of free surface. The water gauge of servo type is used for the measurement of the shape of free surface. The distribution of velocity at the depth $z=2\text{cm}$ measured by using the electromagnetic velocity meter. The velocity of free water surface measured by using the PIV (Particle Image Velocimetry) method. The PIV measurement is a method to determine the velocity by demanding a mean transferring distance of tracer for each measuring point based on a similarity of tracer shape between continuous pictures on the inspection domain. The PVC (Polyvinyl Chloride) powder of mean diameter $50\mu\text{m}$ is used as tracer in this experiment. The PIV analysis is conducted by taking a picture using the video camera positioned at the downstream of channel. A measuring domain is the range of each 50cm in the upstream and downstream side from hydraulic structure. The spatial interval of measurement is 2cm. The program developed by Fujita et al.¹¹⁾ is used for an

analysis of free surface.

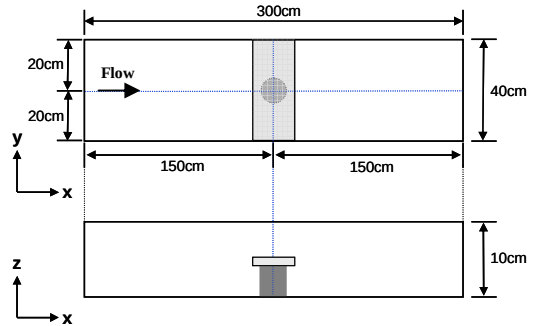
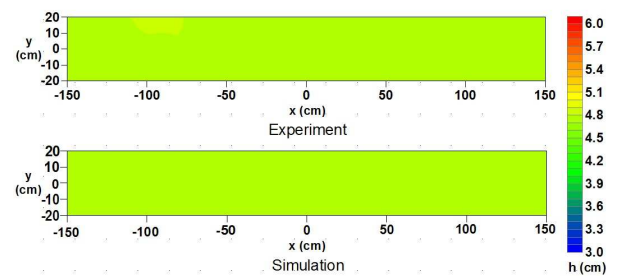
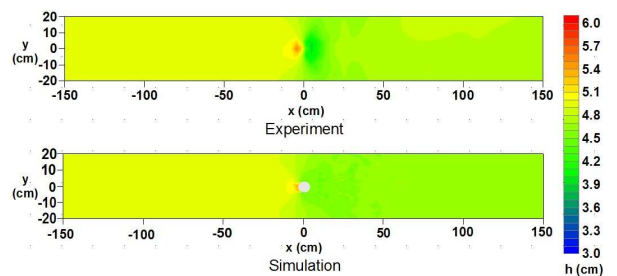
6. RESULTS AND DISCUSSIONS

The simulated results are compared with the experimental data. The computational domain for comparison and the location of structures, shown in Fig 1, are as follows.

The domain for comparison of the water level and the velocity at $z=2\text{cm}$ is the range of each 150cm in the upstream and the downstream side from the hydraulic structure. And, the domain for comparison of the velocity at the free surface is the range of each 50cm in the upstream and the downstream side from the hydraulic structure.

(1) Plane distribution of water level

The results of plane distribution of water level are compared in this section. Fig.2 is the results of Case-1 when there is no structure. Fig.2, Fig.3 and Fig.4 show the results of Case-2, Case-3 and Case-4, respectively. Case-2 is not considering the overtopping flow over hydraulic structures. Case-3 and Case-4 is considering the overtopping flow over hydraulic structures.

**Fig.1** Computational domains**Fig.2** Results of water level (Case-1)**Fig.3** Results of water level (Case-2)

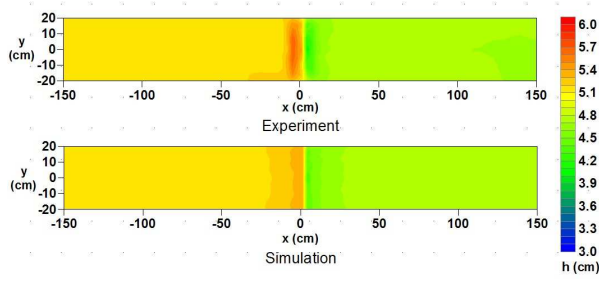


Fig.4 Results of water level (Case-3)

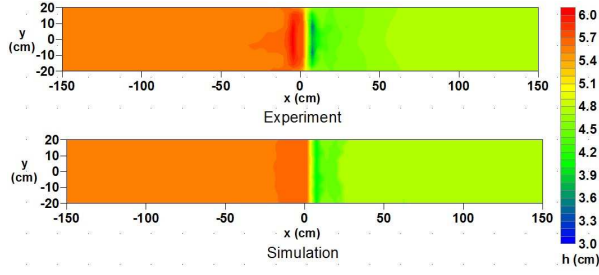


Fig.5 Results of water level (Case-4)

From the computational results, it is judged that the effect of backwater and water level rise by the hydraulic structures generally have good agreements. From the computational results of water level, it is found that tendency of the water level profile can be expressed around hydraulic structures.

(2) Comparison of water level

The water levels along the center line of the flume ($y=0$) of each case are shown in Figs. 6-9, respectively.

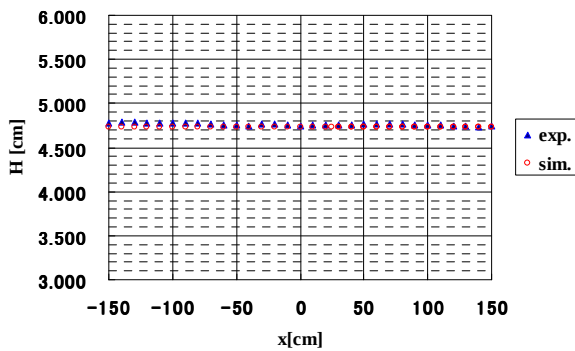


Fig.6 Comparison of water level (Case-1, $y=0$)

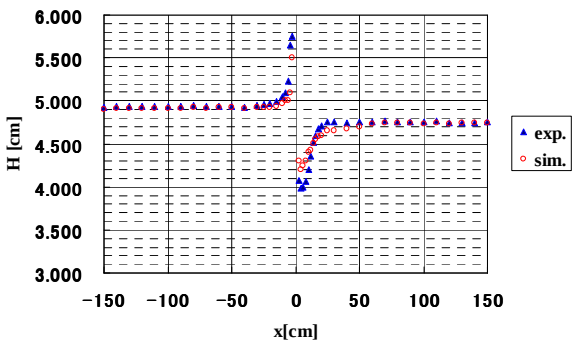


Fig.7 Comparison of water level (Case-2, $y=0$)

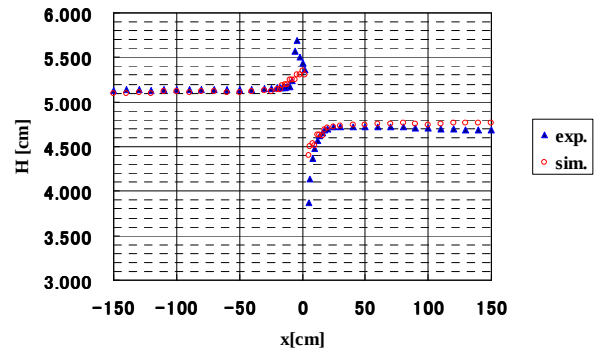


Fig.8 Comparison of water level (Case-3, $y=0$)

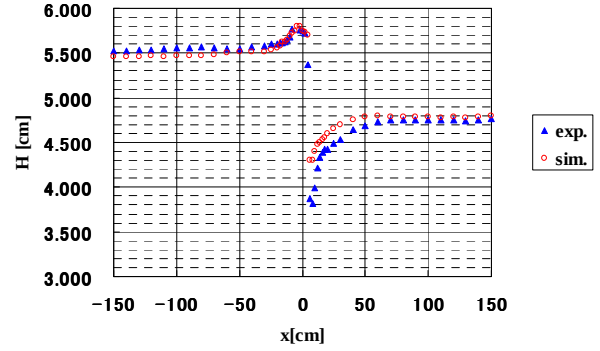


Fig.9 Comparison of water level (Case-4, $y=0$)

From the obtained results, it is found that the water levels generally have good agreements. In general, the effect of backwater by the hydraulic structures was well represented in the simulation.

(3) Comparison of velocity

The simulated velocity at $z=2$ cm is compared with the experimental data in Figs. 10-13. In the figures, x-axis and y-axis show the distance in longitudinal direction and the velocity, respectively.

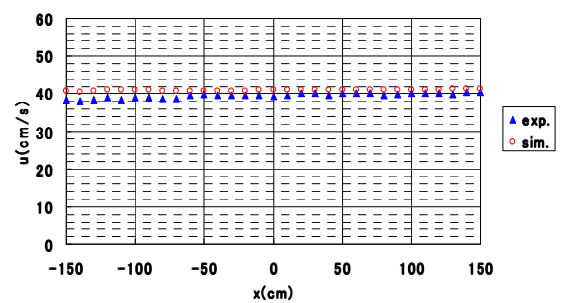


Fig.10 Comparison of velocity at $z=2$ cm (Case-1, $y=0$)

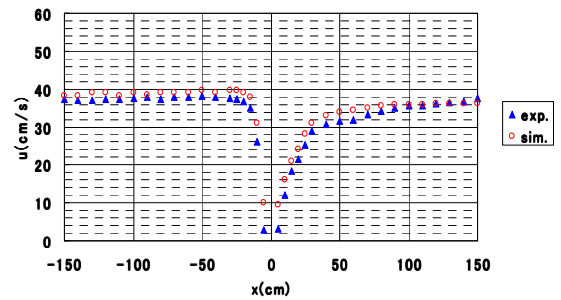


Fig.11 Comparison of velocity at $z=2$ cm (Case-2, $y=0$)

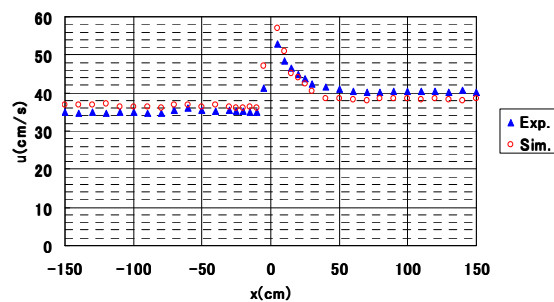


Fig.12 Comparison of velocity at $z=2\text{cm}$ (Case-3, $y=0$)

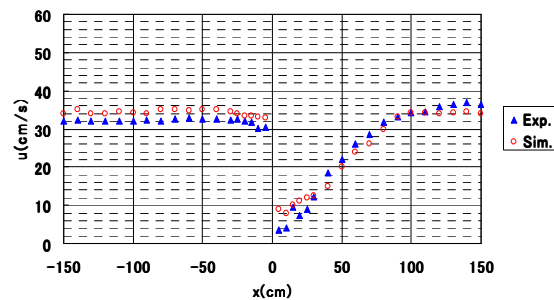


Fig.13 Comparison of velocity at $z=2\text{cm}$ (Case-4, $y=0$)

The above mentioned numerical results of the velocity generally have good agreements with the experimental results. In the numerical results including hydraulic structures, the variations of velocity around hydraulic structures can be seen.

In view of the results so far achieved, it is found that all the results generally can reproduce well the effect of hydraulic structures but the numerical model slightly under-predicts the water level. The main causes of those under-predictions seem to be in the modeling of the turbulence. The turbulence model employed in this study is the standard $k-\varepsilon$ model, which has several problems as pointed out by Speziale¹²⁾, for example the inability to properly account for the streamline curvature, rotational strains and other body force effects and the neglect of the non-local and the effects of the Reynolds stress anisotropies. In order to correct these problems, the consideration of model by introducing non-linear constitutive relation between the mean strain rate and the turbulence stresses is required.

7. CONCLUSIONS

In this study, the numerical simulation was conducted to estimate the effects of water level rise by hydraulic structures within a river. The developed numerical model can treat the flow with free surface on an unstructured mesh with finite volume method. The standard $k-\varepsilon$ model was used for turbulence model and the volume of fluid method proposed by Hirt and Nichols¹⁾ was used to

represent the free water surface. The differencing scheme proposed by Ubbink and Issa⁵⁾ is also employed to capture the free water surface in unstructured mesh.

The prediction of the water level rise by hydraulic structures is very important from the viewpoint of flood disaster. The present study shows that the numerical model used in this study can be used to simulate the changes of the flow field around hydraulic structures although the numerical model underestimates the water level rise around hydraulic structures. In order to improve the model developed in this study, further researches considering different turbulence models and various flow conditions are needed.

REFERENCES

- 1) Hirt, C. W. and Nichols, B. D.: Volume of fluid (VOF) method for dynamics of free boundaries, *Journal of Computational Physics*, Vol.39, pp.201-225, 1981.
- 2) Ashgriz, N. and Poo, J. Y.: Flux line-segment model for advection and interface reconstruction, *Journal of Computational Physics*, Vol.39, pp.449-468, 1991.
- 3) Lafauie, B., Nardone, C., Scardovelli, R., Zaleski, S. and Zanetti, G.: Modelling merging and fragmentation in multiphase flows with SURFER, *Journal of Computational Physics*, Vol.113, pp.134-147, 1994.
- 4) Ubbink, O.: Numerical prediction of two fluid systems with sharp interfaces, *Ph.D. thesis, University of London*, 1997.
- 5) Ubbink, O. and Issa, R. I.: A method for capturing sharp fluid interfaces on arbitrary meshes, *Journal of Computational Physics*, Vol.153, pp.26-50, 1999.
- 6) Rodi, W.: Turbulence models and their application in hydraulics-a state of the art review, *University of Karlsruhe, Karlsruhe, Germany*.
- 7) Rhie, C. M. and Chow, W. L.: A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation, *AIAA Journal*, Vol.21, pp.1525-1532, 1983.
- 8) Saad, Y.: Iterative methods for sparse linear system (second edition), *Society for Industrial & Applied Mathematics*.
- 9) Patankar, S. V.: Numerical heat transfer and fluid flow, *McGraw-Hill, New York*.
- 10) Zhang, H., Nakagawa, H., Ishigaki, T. Muto, Y.: A RANS solver using a 3D unstructured FVM procedure, *Annals of Disas. Prev. Res. Inst., Kyoto University*, No.48 B, 2005.
- 11) Fujita I., Muste M., and Kruger A., Large-scale particle image velocimetry for flow analysis in hydraulic engineering applications, *Journal of Hydraulic Research*, Vol. 36, No. 3, pp. 397-414, 1998.
- 12) Speziale C. G., Analytical methods for the development of Reynolds-stress closures in turbulence, *Annual Review of Fluid Mechanics*, Vol. 23, pp. 107-157, 1991.

(Received September 30, 2009)