# Ⅱ－12 DEVELOPMENT OF THREE DIMENSIONAL NUMERICAL MODEL FOR A TSUNAMI IN A OPEN-CHANNEL WITH OBSTACLES

Tohoku University Student member O Sung Jin HONG
Tohoku University Member Fumihiko IMAMURA

## 1. Introduction

Numerical models are tried to be developed for analyzing the resistance force interacted between wave and structures and applied with sufficient accuracy to practical problems, In the computation, however, it is normally assumed either that there are no solid structures capability of withstanding the tsunami, or that the effect of a solid structure can be expressed in terms of a roughness coefficient, such as Manning's *n*, which is selected without/with due consideration of the hydraulic characteristics. Since the effect of obstacles is important for disaster prevention of tsunami in shallow water and on land, it is necessary to analyze the resistance force interacted between wave and structures. However, two-dimensional numerical model with Manning's *n* which is selected without/with consideration of the hydraulic characteristics could not represent the interaction between wave and structures because that model could not analyze exactly the effect of structure for tsunami (Aburaya & Imamura, 2002; Hong & Imamura, 2003). In this study, therefore, three-dimensional numerical tsunami model try to be developed to analyze the nonlinear behavior of flow around obstacles based on the Navier-Stokes equations.

## 2. Numerical Approximation

Three-dimensional flow of a viscous incompressible fluid is considered in the model. The non-dimensional govering equations used in the present model are the continuity and momentum equations of the Navier-Stokes equations as follows:

$$\nabla \cdot u = 0$$
$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla p + \nu\nabla^2 u + g \qquad (1)$$

in where $p$ is the ratio of the pressure to the constant density and $u$ is the velocity vector. The kinematic viscosity coefficient designated by $\nu$. The body force is designated by the constant acceleration components $g$. The MAC method (Harlow & Welch, 1965; Hirt & Cook, 1972) employs an Eulerian mesh at calculational cells and finite difference expressions to solve the governing equations numerically. The primary dependent variables are the pressure and the velocity components of the fluid. The pressure is calculated through the solution of a Poisson equation for pressure. A staggered mesh is used for the velocity component, which specifies the normal velocities at the Eulerian cell boundaries (see Fig. 1). In numerical approximation, the central difference scheme with the error of second-order is used to the space derivative. But, since numerical approximation based on the centered differences have been unstable, an upwind scheme with error of second order in convection terms of Eq. (1) is employed in order to make the computation stable.

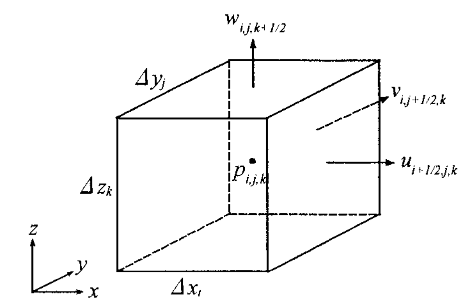



**Figure 1**. Positions of velocity and pressure components on a typical Eulerian cell $(i,j,k)$

## 3. Validation Test Case

A simple problem is selected as test model for comparison, which is an ideal test case for investigation of the propagation of internal flow through a boundary. The computational region for the test calculation is shown in Fig. 2.

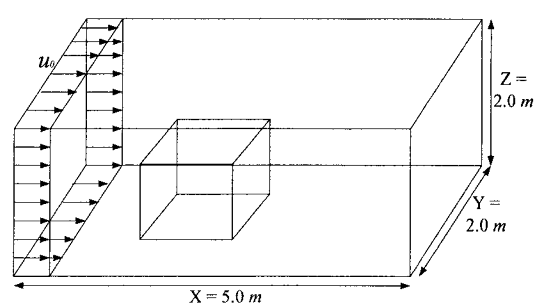
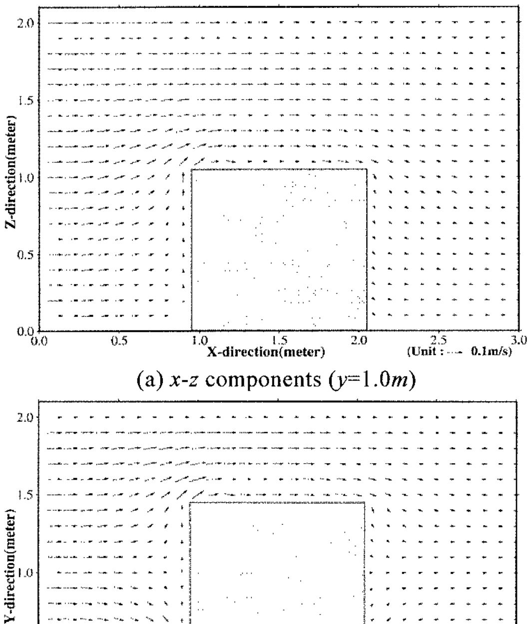


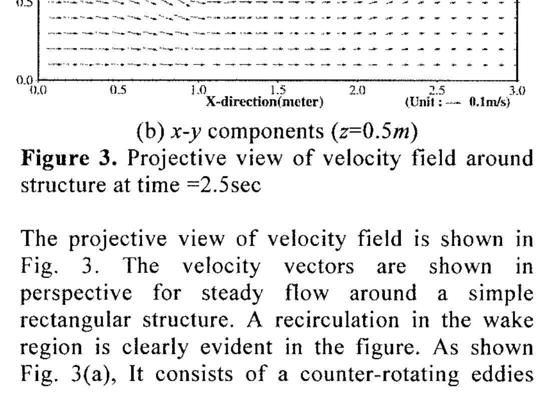

**Figure 2**. A schematic sketch of the computational domain

The origin of $x$ coincides with boundary for uniform flow. The positive direction $x$-axis is taken toward the right hand side of Fig. 2. We assumed that there are no accelerations due to gravity ($g$). The computational grid cells have dimensions of 0.1m in the $x$, $y$ and $z$ direction. The computational region was divided in to 50 meshes in the $x$-direction, 20 meshes in the $y$-direction and 20 meshes in the $z$-direction. A time-step of 0.025s, determined from the stability conditions, was used in the calculation. In this study, the MAC method is employed for analyzing the governing equations numerically; however, it is known that the calculation results by the MAC method depend on the initial velocity field. Therefore, it is important to use appropriate velocity distributions for the initial conditions. But, for test calculation, we set the boundary condition for the flow as follows:

$$u_0 \begin{cases} = 0.1\dfrac{t}{0.25}(0 < t < 0.25\text{sec}) \\ = 0.1 \qquad (t \geq 0.25\text{sec}) \end{cases}, v_0 = w_0 = 0.0\ (m/s) \quad (2)$$



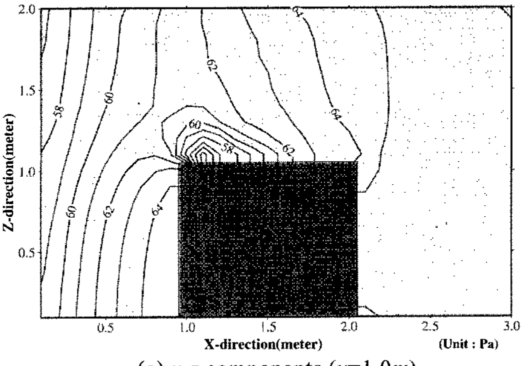

(a) $x$-$z$ components ($y$=1.0$m$)



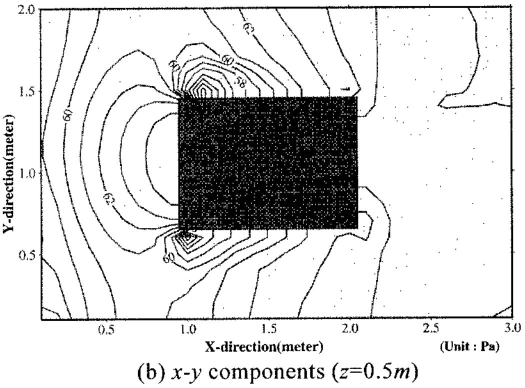

(b) $x$-$y$ components ($z$=0.5$m$)

**Figure 3.** Projective view of velocity field around structure at time =2.5sec

The projective view of velocity field is shown in Fig. 3. The velocity vectors are shown in perspective for steady flow around a simple rectangular structure. A recirculation in the wake region is clearly evident in the figure. As shown Fig. 3(a), It consists of a counter-rotating eddies that are large near the top of the structure, but small near its base. Also, the double eddy structure is clearly seen in Fig. 3(b).




(a) $x$-$z$ components ($y$=1.0$m$)




(b) $x$-$y$ components ($z$=0.5$m$)

**Figure 4.** Projective view of pressure field around structure at time =2.5sec

The distribution of pressure is shown in Fig. 4. As shown this figure, the pressure is decreased near the top of the structure and near its side; it is increased near the front of the structure. Such distribution of pressure, in the side and top of the structure, is presumably cause by large counter-rotating eddies around structure.

Since the numerical model has not been developed to estimate the resistance force interacted between wave and building/structure, the three-dimensional numerical model has been developed using MAC (Marker and Cell) method and carried out about simple problem.

## References


Harlow, F.H. and Welch, J.E. (1965), "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", *Phys. Fluids*, Vol. 8, No. 12, pp 2182-2189.

Hirt, C.W. and Cook, J.L. (1972), "Calculating Three Dimensional Flows around Structures and over Rough Terrain", *Journal of Computational Physics*, Vol. 10, pp. 324-340.