

# Numerical Analysis of Ocean Wave by Multi-Grid Method

*S. H. Kwag\**

복합격자 방법에 의한 해양파의 수치해석

곽 승 현

**Key Words** : Multi-Grid(복합격자), Free-Surface(자유표면), Viscous Flow(점성흐름), Triple Mesh Method(삼중격자법), Navier-Stokes Equation(나비에스톡스 방정식)

## Abstract

The ocean wave is hydrodynamically investigated to get more reliable solution. To improve the computational accuracy, more fine grids are used with relatively less computer storage on the free surface. One element of the free surface is discretized into more fine grids because the free-surface waves are much affected by the grid size in the finite difference scheme. Here the multi-grid method is applied to confirm the efficiency for the S103 ship model by solving the Navier-Stokes equation for the turbulent flows. According to the computational result, approximately 30% can be improved in the free surface generation. Finally, the limiting streamlines show that the numerical result is similar to the experiment by twin tuft.

## 1. Introduction

To overcome the deficiency of computer's hardware, the improvement of the efficiency has been strongly demanded in the computational procedure. Many numerical techniques have been developed to improve the finite difference method, but the method still faces a serious problem

because it requires very long CPU time and a huge memory storage for accurate simulation. The method of IAF (implicit approximation factorization), a kind of implicit scheme, the method of local time step, etc. are the examples for more efficient computations. Some comparative calculations by these methods have been carried out. It seems that IAF is

---

\* 정회원, 한라대학교 조선공학과 교수

quite promising to speed up the calculation but its formulation is a little complicated. Moreover, the method brings forth some numerical damping. For the numerical truncation error to be small enough to have little effect on the physical performance, the mesh size should be strictly considered because it is very crucial for the efficiency. For example, in the finite difference calculations of viscous flow described by the Navier-Stokes equation, the minimum mesh size is usually so chosen that the numerical dissipation, which comes from the discretization of the convection terms, is much less than the physical dissipation. The mesh size must be extremely small for high Reynolds-number flows to meet this demand. However, such fine meshes are not always necessary for all the equations and terms. For example, the truncation errors of the Poisson equation for the pressure of the non-convective terms in Navier-Stokes equation do not have much influence on the results as the convection terms do. The hybrid type of the mesh may make the computations more efficient. One possibility is to employ different mesh systems depending on the characteristics of the equations or the terms. We call such a method "double mesh method<sup>1)</sup>" or "triple mesh method<sup>2)</sup>", written in short as DMM or TMM hereafter. It was first proposed for numerical simulations of 3-D nonlinear free-surface flow problems by boundary element method<sup>3)</sup>. In order to reduce the numerical viscosity as much as possible, a very fine mesh system which contains about 60 grids<sup>4)</sup> in one wave length is used in the finite difference calculation concerned with the free-surface equations, while the governing Laplace equation is solved on a relatively coarse

mesh system which contains about 10 grids in one wave length by the boundary element method. The computed results by DMM or TMM were of enough accuracy and both the computing time and the size of the memory storage were remarkably reduced.

In the present study, a multi-grid on the free-surface is introduced in the finite difference solver of the Navier-Stokes equation to improve the calculation efficiency. As mentioned above, the demands to the mesh size are not the same for all the equations and the terms in the finite difference method. So it is expected that some improvement, similar to that achieved in the simulation of free-surface problem by DMM or TMM, may be made by introducing more fine meshes in the conventional finite difference method.

## 2. Governing Equations and Computational Strategy

A single grid system<sup>5)</sup> is usually used in the whole computation whose minimum size is determined for the numerical diffusion to be less than that by viscosity. The grid size for the calculation of the free surface elevation must be determined by a different scale, the minimum wave length. In the simulation, two or three mesh systems are usually used whose sizes are different each other depending on the characteristic of equations. The first one is for the convective terms in the Navier-Stokes equation, the second is for the Poisson equation, and the third is for the free-surface equation. The third grid system requires the finest mesh. In the present calculation, the third one is numerically

confirmed ; more fine grids are used to improve the accuracy of free-surface calculation with relatively less computer storage. One element of the free-surface is discretized into (4xii,4xjj), (8xii,4xjj), (12xii, 4xjj) fine grids because the free-surface waves are much affected by the grid size in the finite-difference scheme.

The positions, or Lagrangian coordinates, of each particle  $(x_p^n, y_p^n, z_p^n)$  are obtained by numerical integration from some initial position  $(x_p^0, y_p^0, z_p^0)$  at time = 0 ;

$$\begin{aligned} x_p^n &= x_p^0 + \int u_p \cdot dt \\ y_p^n &= y_p^0 + \int v_p \cdot dt \\ z_p^n &= z_p^0 + \int w_p \cdot dt \end{aligned} \quad (1)$$

where  $u_p, v_p, w_p$  are the velocities in the Eulerian mesh at the time dependent location of the particle. In the present MAC-based codes, the particle velocities are evaluated by two-variable linear interpolation. Consistent with the forward time integration of MAC method, (1) is evaluated sequentially as (2).

$$\begin{aligned} x_i^{n+1} &= x_i^n + u_i^n \cdot \Delta t \\ y_i^{n+1} &= y_i^n + v_i^n \cdot \Delta t \\ z_i^{n+1} &= z_i^n + w_i^n \cdot \Delta t \end{aligned} \quad (2)$$

(2) is the Lagrangean expression of the kinematic condition on the free-surface. The condition can also be expressed in the Euler form as follows:

$$\partial \zeta / \partial t = - u \cdot \partial \zeta / \partial x - v \cdot \partial \zeta / \partial y + w \quad (3)$$

where  $\zeta$  and  $t$  are the free-surface elevation and the time respectively. Numerically (2) is equiv-

alent to (3) if the 1st order upstream difference scheme is used in (3).

The shape of the free-surface is not known a priori; it is defined by the position of the marker particles. We note that the boundary conditions at the free-surface require zero tangential stress and a normal stress which balances any externally applied normal stress. The application of these conditions requires a knowledge of not only the location of the free-surface at each grid but also its slope and curvature. In our calculation, the  $z$ -coordinate of the free surface is re-arranged by the bivariate linear interpolation in proportion to the newly calculated projected area at each time step. At the hull surface, the no-slip condition is used for the velocity, and the Neumann condition for the pressure. The hull is fixed in the computational domain, while the uniform flow is imposed on the upstream boundary.

### 3. Application and Discussion

The S-103 model for high Reynolds-number flows is studied to confirm the numerical efficiency of the multi grid. S-103 is an Inuid model with the beam/length ratio of 0.09. In the present case, calculations are made at  $R_n=10^6$  and  $F_n=0.28$  with Baldwin Lomax turbulence model. The result is that at the time  $T=3.0$ , when the convergence is well assured. The grid size of regular type is 74x29x30 and the multi-grid on the free-surface is numerically tried. Fig. 1 shows the wave patterns obtained by the regular grid. Fig. 2 uses the grid of (4xii,4xjj) on free surface and gives us about 7% improvement in the free-surface development, compared with that

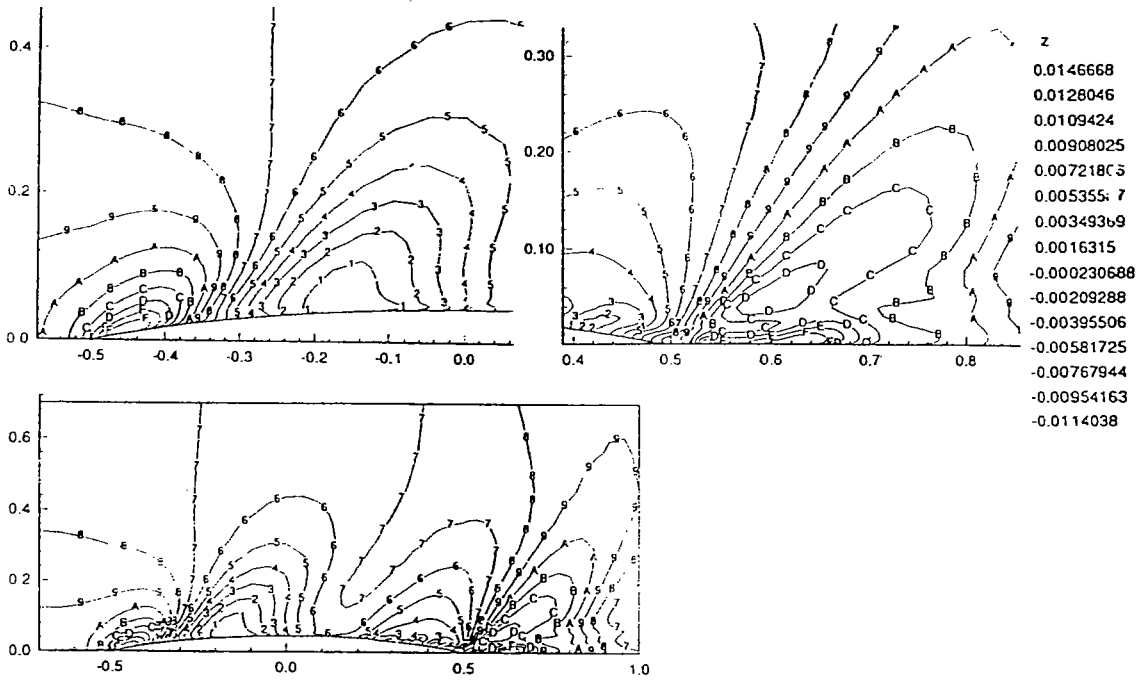


Fig. 1 Free-surface contour by regular grid(ii, jj) for S-103 case

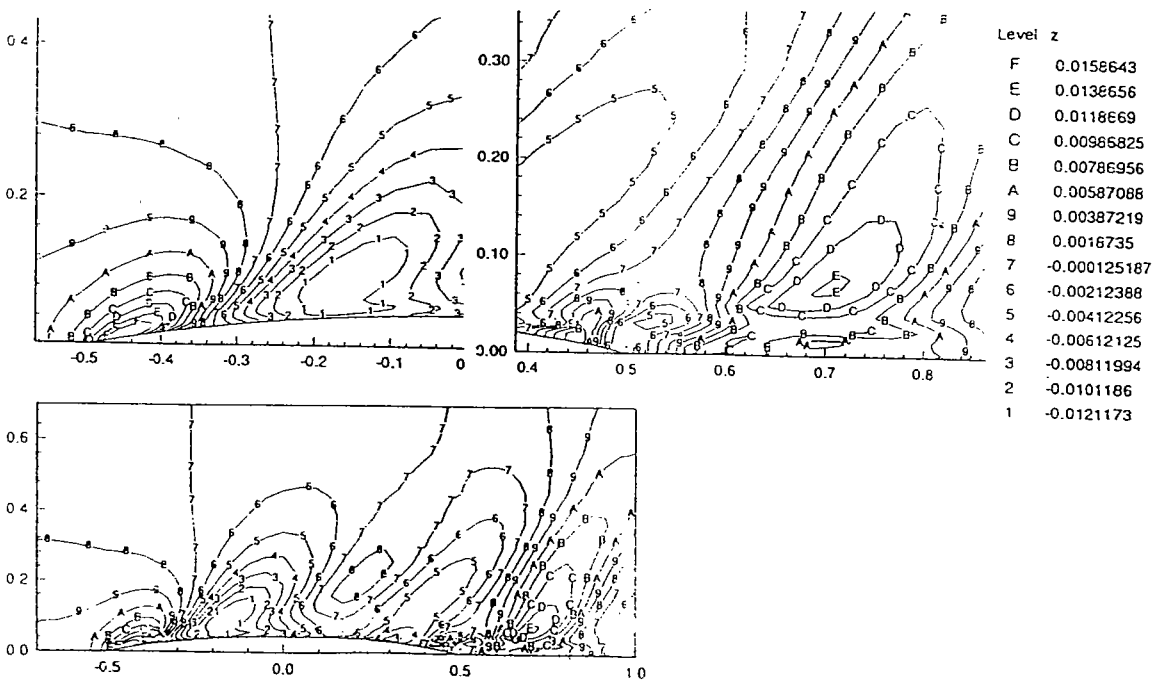


Fig. 2 Free-surface contour by multi-grid (4xill, 4xjj) for S-103 case

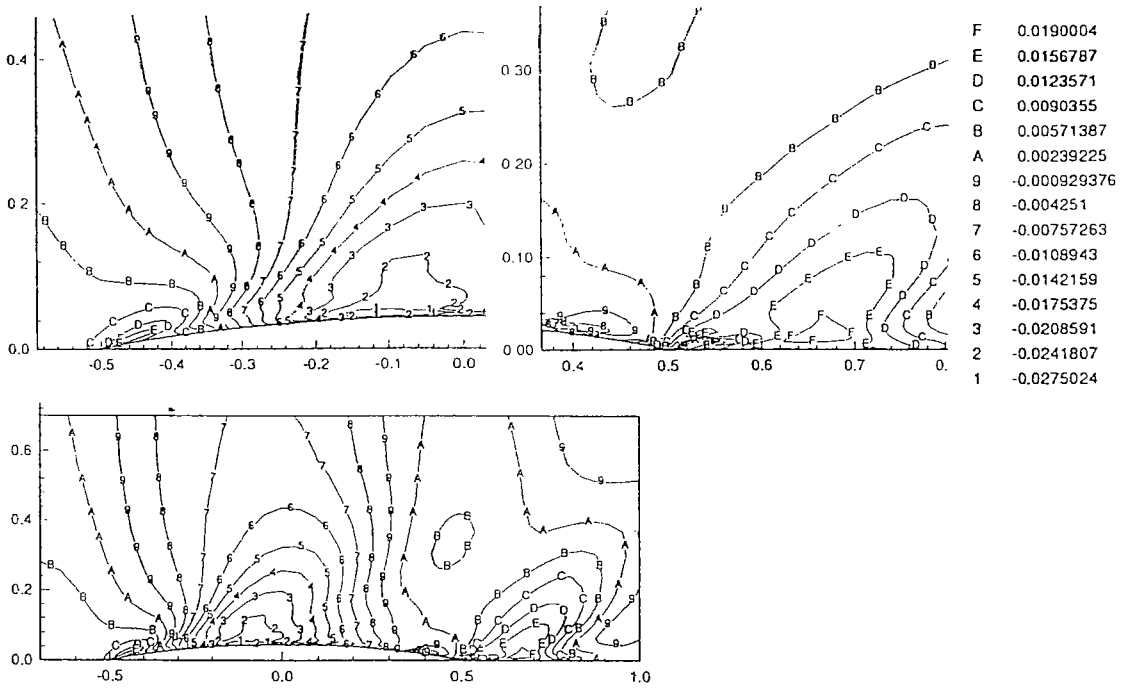


Fig. 3 Free-surface contour by multi-grid (8x11, 4xj) for S-103 case

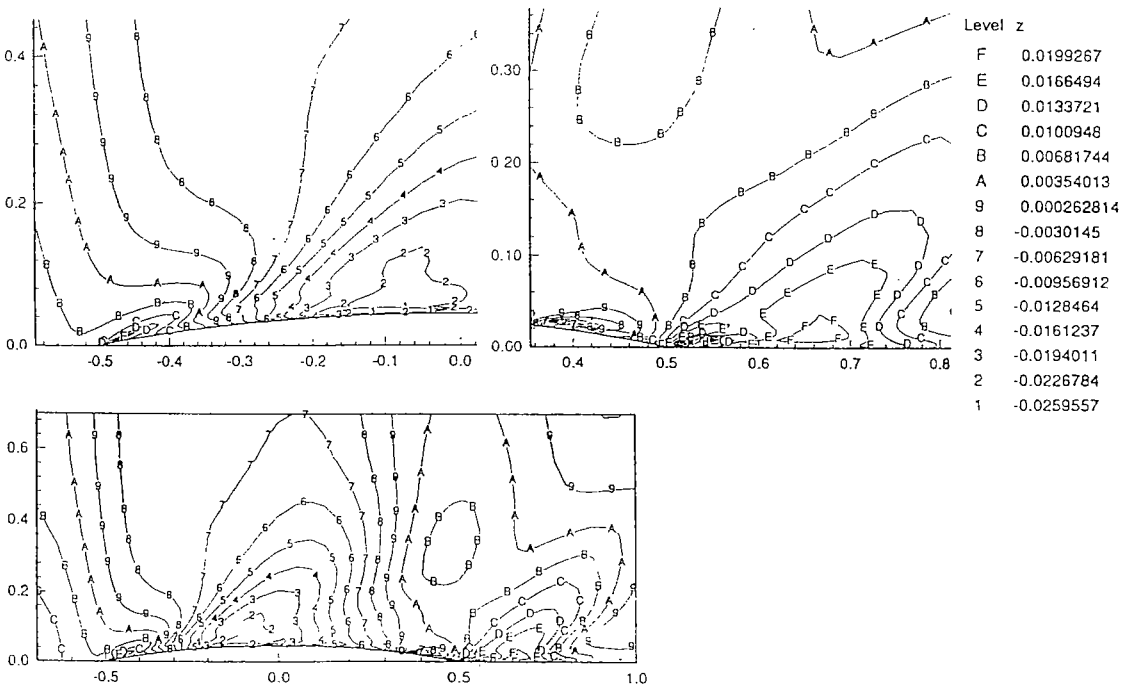


Fig. 4 Free-surface contour by multi-grid (12x11, 4xj) for S-103 case

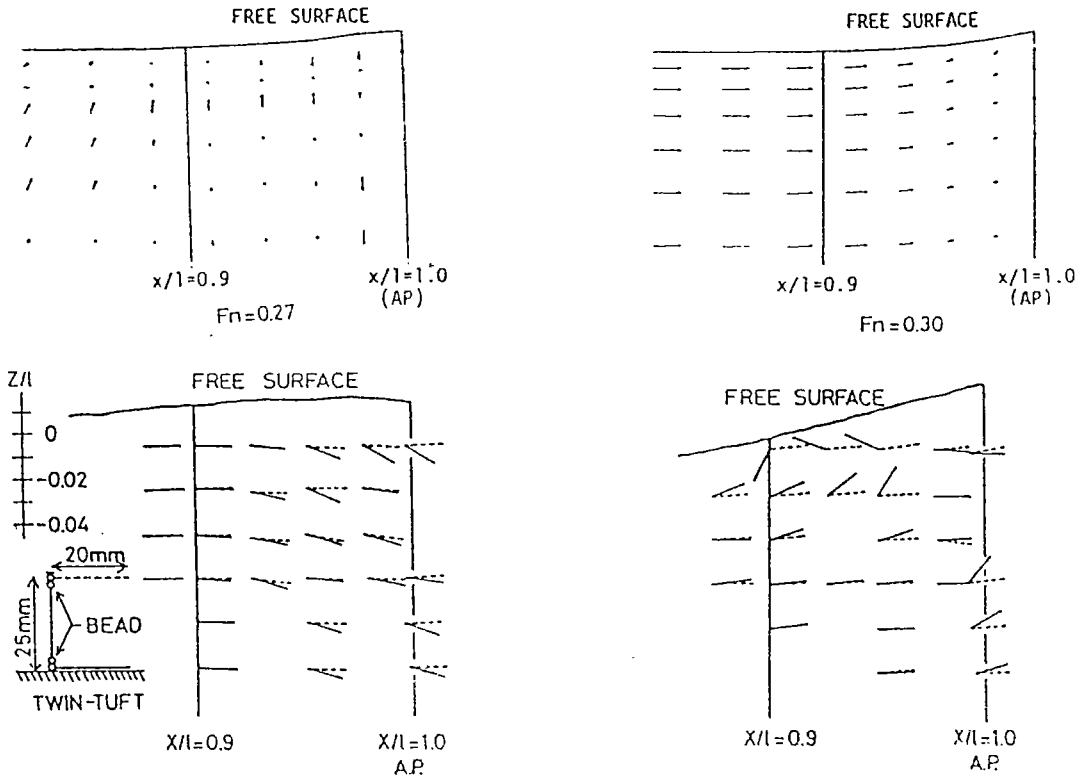


Fig. 5 Limiting streamlines at  $Fn=0.27$  and  $0.30$ (above:computed, below:experiment)

by regular grid. It is found that the size of grid is very important in moving the marker particles on free surface. Here the improvement means that the wave height, i.e., the distance between the trough and the crest, could be numerically calculated and quantitatively compared to each other.

Fig. 3 uses that of  $(8x_{ii}, 4x_{jj})$  on free-surface. It shows the improvement of about 21% at  $T=3.0$ . Fig. 4 uses that of  $(12x_{ii}, 4x_{jj})$  on free-surface. Some more improvement is obtained; 29% compared with that of regular grid at  $T=3.0$ .

Fig. 5 shows the limiting streamlines. Here the solid line means the streamline in the boundary

layer and the dotted line outside it. The separated region of  $Fn=0.27$  is significantly wider than that of  $0.30$ . The experiment by twin tuft shows similar tendency; the separated region of  $Fn=0.30$  close to the free surface is partially due to the free-surface sub-breaking.

#### 4. Conclusion

The multi-grid method is applied to the finite difference solution of the Navier-Stokes equation to improve the efficiency. The method is to use more than one mesh system on the free surface.

Through comparative computations, the method is found to be significantly effective for the efficient free-surface calculation without much increase of CPU time and the memory. Finally, the limiting streamlines show that the numerical result is similar to the experiment by tufts.

### 요 약

해양파 해석에서 자유표면의 효율을 향상시키기 위하여 복합격자법을 사용, 점성유동장 계산을 수행하였다. 지배방정식은 Navier-Stokes 방정식이고 난류모형은 Baldwin-Lomax이다. 지배방정식의 각항에 서로 다른 크기의 격자를 적용, 계산기의 용량을 최소로 유지하면서 계산시간은 단축하고, 정도를 최대한 향상하였다. 대상선형은 S-103 선박이다.

### References

1) Xu, Q., Mori, K. and Shin, M., Double Mesh

Method for Efficient Finite Difference Calculation, J. of Soc. of Naval Arch. of Japan, Vol. 166, 1989.

- 2) Mori, K., Kwag, S. and Doi, Y., Numerical Simulation of Ship Wave and Some Discussions on Bow Wave Breaking and Viscous Interaction of Stern Wave, 18th Symp. on Naval Hydrodynamics, U.S.A., 1990.
- 3) Xu, Q. and Mori, K., Numerical Simulation of 3-D Nonlinear Water Wave by Boundary Element Method, J. of Soc. of Naval Arch. of Japan, Vol. 165, 1989.
- 4) Shin, M. and Mori, K., Numerical Computation of 2-D Waves behind a Hydro foil, J. of Soc. of Naval Arch. of Japan, Vol. 163, 1988.
- 5) Kwag, S. H., Mori, K. and Shin, M., Numerical Computation of 3-D Free Surface Flows by N-S Solver and Detection of Sub-Breaking, J. of Soc. of Naval Arch. of Japan, Vol. 166, 1989.