論 文

Transactions of the Society of Naval Architecture of Korea
Vol. 31, No. 4, November 1994
大 韓 造 船 學 會 論 文 集
第 31 卷 第 4 號 1994年 11月

A Study on Grid Dependencies of the Numerical Solutions for Ship Viscous Flows

by

K. J. Kang* and S. H. Lee**

배주위 점성유동장에 대한 수치해의 격자의존성에 관한 연구

강국진*, 이승희**

Abstract

It is very important to understand characteristics of solution due to the variation of computational grid sizes, especially when turbulence model not incorporating wall-function is used. The present paper performs numerical investigation on the grid dependency of numerical solution for three dimensional turbulent flow field around a ship. In the present study a finite volume method with a modified sub-grid scale turbulence model and a numerically constructed non-orthogonal curvilinear coordinate system capable of conforming complex ship geometries are used. Numerical studies are then performed for a mathematical Wigley hull and the Series 60, C_B=0.8 hull forms.

The results for various grid sizes are compared with each other and with measured data to show grid dependencies of numerical solutions.

요 약

계산격자크기의 변화에 따른 해의 특성을 이해하는 것은 중요하며, 특히 벽함수를 사용하지 않는 난류모형을 사용할 때 더욱 그러하다. 본 논문에서는 배주위의 3차원 난류유동장에 대한 수치해의 격자의존성에 대한 수치계산적인 조사를 수행한다. 본 연구에서는 수정된 sub-grid-scale 난류모형과 함께 유한체적법을 사용하며 복잡한 배의 기하학적 형상에 적합한 비직교의 곡선좌표계를 수치적으로 만들어 사용한다. 그리고 수학선형인 Wigley 선형과 Series 60 (C_B=0.8) 선

발 표 : 1993년도 대한조선학회 춘계연구발표회('93. 4. 17.)
Manuscript received : September 27, 1993, Received manuscript received : October 10, 1993
* Member, Korea Research Institute of Ships & Ocean Engineering
** Member, Dept. of Naval Architecture & Ocean Engineering, Univ. of Inha

형에 대하여 수치계산적인 연구를 수행하고, 수치해의 격자의존성을 보이기 위하여 여러가지 격자 크기에 대한 계산결과들을 서로 비교하였으며 실험결과와도 비교해 보인다.

1. INTRODUCTION

The flow field around a ship, in general, is very complicated due to its geometrical shape and presence of free-surface wave system. Near the bow, the flow is almost ideal and various theoretical approaches applicable to the practical design of ship bow are now available.

However the flow field near the stern is subjected to the strong effects of viscosity and the assumptions of ideal flow are no longer valid in this region. The stern flow field is of primary interest to naval architects since, for example, usually the core of longitudinal vorticies is present on the propeller plane and significantly affects the efficiency of ship propulsion.

Therefore, numerous experiments and numerical computations have been performed to analyse the complicated stern flows. However, as discussed at the second SSPA-CTH-IIHR Workshop[1], the agreements between measurements and computations for HSVA tankers are not satisfactory. Almost all of the results presented at the workshop failed to accurately predict the vortex intensities, but the reason was not clarified. Since, as is well known to numerical experimenters, failures in accuracies can occur due to any combination of the following factors; use of averaging procedures, insufficient turbulence model, insufficient grid resolution and etc.

Maintaining sufficient grid resolutions for high Reynolds number flows are impracticable and it is customary to use a turbulence model.

The two-equation k-ε model with wall-function and algebraic Balwin and Lomax model with no slip boundary conditions[2] are most frequently used. Sub-grid scale (SGS) turbulence model is also used [3,4,5] but this model is strongly dependent on grid resolutions and usually require finer grid spacings at near-wall region, causing longer computational time than other turbulence models.

Authors[4,5] have already developed a numerical method for the simulations of three dimensional turbulent flow field around a ship. The method employs non-orthogonal curvilinear coordinate system and a finite volume method. The Navier-Stokes equations are discretised in a staggered grid system and a SGS model is used for approximations of flow turbulences.

The goal of the present study is to study the effects of grid resolutions of the numerical results and, by doing so, to find out appropriate grid sizes which can give sufficient accuracy for prediction of wake field, usable in practical ship design. Kodama[6,7] also have reported the influences of variation in the minimum grid spacing to the numerical result.

The numerical computations with various grid sizes are performed for a mathematical Wigley and the practical Series 60, $C_B = 0.8$ hull forms and are verified by comparing the results with measured data[8,9]. The optimum grid size is found and it is demonstrated that the present numerical approach with an optimum grid is capable of predicting ship stern flow characteristics accurately enough for primary ship design.

2. FORMULATION

2.1 Governing equations

For an incompressible flow field around a three dimensional body, the dimensionless continuity equations and the Navier-Stokes equations transformed from Cartesian coordinates x_i to a general coordinates ξ^i, while maintaining Cartesian velocity component u_i, can be written as follows if summation notation is used:

$$0 = \frac{1}{J}\frac{\partial}{\partial \xi^i}(A_k^i u_k) \tag{1}$$

$$\frac{\partial u_i}{\partial t} = \frac{1}{J}\frac{\partial}{\partial \xi^j}(-A_k^j p \delta_{ik} + A_k^j \sigma_{ik}) \tag{2}$$

$$\sigma_{ik} = -u_i u_k + \frac{2}{R_n} e_{ik} - \overline{u_i u_k} \tag{3}$$

$$e_{ik} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i}\right) \tag{4}$$

$$R_n = \frac{U_o L}{\nu} \tag{5}$$

and A_j^i is the j-th Cartesian components of A^i, the area vector in ξ^i-direction, $i.e.$,

$$A_j^i = J\frac{\partial \xi^i}{\partial x_j} \tag{6}$$

which is equivalent to Jacobian J times x_j-component of the contravariant base vector a^i.

2.2 Turbulence model

Since the direct computation of turbulent flow field using above equations is practically impossible, it is customary to use some kind of approximations, i.e., turbulence model. In the present paper, the modified SGS model proposed by Miyata[3] is adopted. Original SGS model is used in the large eddy simulations and has simple equations with the least number of empirical constants. This model is expected to realize more realistic simulation of physical phenomena. However, the flow upstream is uniform without turbulence and it interacts with a ship in the after region, resulting in the generation of a viscous turbulent flow. Therefore the fluctuating part of the velocity cannot be calculated by the procedure used in the so-called "large-eddy simulation" technique. And the application is limited to the simple problem at the rectangular grid system since the grid sizes need to be nearly homogeneous. However, in the case of ship, the grid spacing is not sufficiently fine for this treatment. Then only the eddy viscosity of this model is incorporated in the calculation. In this sense the modified SGS model is used in an averaged manner.

The SGS Reynolds stresses are approximated with the Boussinesq's eddy viscosity concept as follows :

$$\overline{u_i' u_j'} = \frac{2}{3}\frac{\nu_s^2}{(0.094\triangle)^2} - 2\nu_s \overline{e_{ij}} \tag{7}$$

where filter width $\triangle$ is a characteristic grid scale defined as

$$\triangle = (\triangle x_1 \triangle x_2 \triangle x_3)^{\frac{1}{3}} \tag{8}$$

The Smagorinsky[10] assumption for the eddy viscosity ν_s is derived as

$$\nu_s = L_s^2 (2\overline{e_{ij}}\,\overline{e^{ij}})^{\frac{1}{2}} \tag{9}$$

where the length scale is set at half of the minimum grid spacing following Takakura [11] :

$$L_s = 0.5 \min(\triangle x_1, \triangle x_2, \triangle x_3) \tag{10}$$

3. NUMERICAL METHOD

To develop a numerical method for simulations of the turbulent flow fields around ships, a finite volume method is adopted and equations are discretised in a staggered grid system. Pressures are defined at the centers of the control volumes and velocity components at the centers of the corresponding control surfaces. An algebraic grid generation technique incorporates with an elliptic grid generation technique is used for the constructions of the non-orthogonal grid system conforming complex hull geometries. Schematic diagrams on the relation of physical and transformed computational domains are shown in the Fig. 1.

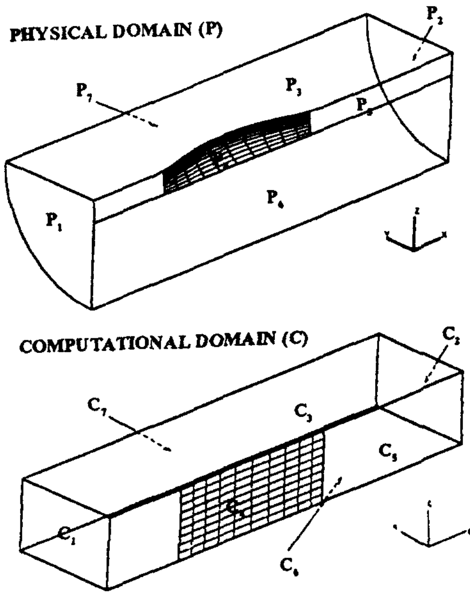


Fig. 1 Transformation of the Coordinate Systems

3.1 Grid generation

Curvilinear grid system conforming the given ship geometries is numerically generated by an algebraic grid generation technique[4]. To ensure the smoothness of the resulting grid system, however, few sweeps of the grid by an elliptic grid generation technique(Thompson [12]) may be necessary. Since H-H type grid topologies are not easily applicable to the ships with flat bottom which consists most of the commercial vessels, O-H type grid systems are chosen in the present study. The numerically generated coordinate system for the Wigley and the Series 60, C_B=0.8 hull form is shown in the Fig. 2.

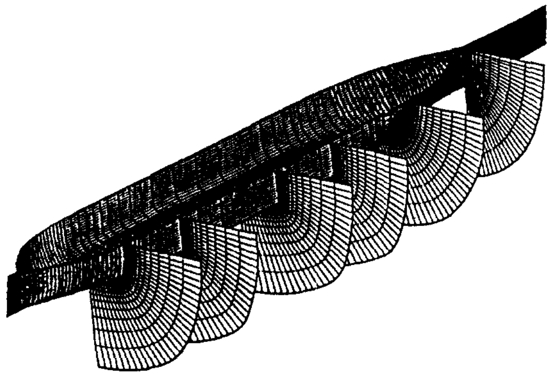


Fig. 2 Numerical Grids generated for Series 60 (C_B=0.8)

The maximum of 78,000 (97×35×23) grid points for the Wigley hull and about 126,480 (124×34×30) points for the Series 60, C_B=0.8 hull form are used. The minimum grid spacing close to the hull surface is determined so that $y^+ = \frac{y u_\tau}{\nu}$, where u_τ is the friction velocity, do not exceed 10.

3.2 Numerical method

The unsteady incompressible three-dimensional NS equations discretised in the generalized curvilinear coordinate system are then solved by the time marching procedure where the convective terms are approximated by the QUICK (Leonard[12]) scheme, all the other spatial derivatives by the second order centered differencing and temporal derivatives by the first order forward differencing. Velocity and pressure fields are simultaneously iterated as suggested by Hirt et al.[14]. Details on the numerical methods can be found Kang[4].

3.3 Boundary conditions

Boundary conditions imposed in the present study for the calculations of flow field around a ship are summarized in the Fig. 3.

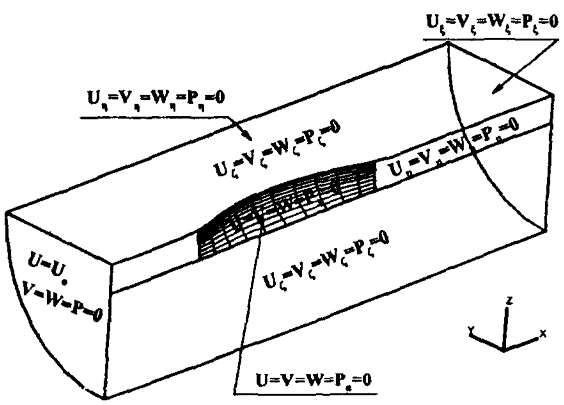


Fig. 3 Boundary Conditions for Numerical Computations

Free-surface is neglected and the flows about double bodies are simulated with symmetric boundary conditions. No-slip and zero pressure gradient conditions are enforced at the hull surface. At the up and downstream boundaries, gradient of the u_i and p are set to zero. Entire flow regions are assumed turbulent when SGS model is used.

4. RESULTS

4.1 Wigley Hull

The shape of the Wigley hull are defined by the equation given below :

$$y = \frac{B}{2}\left\{1-\left(\frac{2x}{L}\right)^2\right\}\left\{1-\left(\frac{z}{d}\right)^2\right\} \quad (11)$$

where y is the offset of the hull and L, B and d are length, breadth and draft of the ship, respectively. The length-to-breadth ratio is set at 10 and length-to-draft ratio at 16.

Turbulent flow fields around the Wigley hull at $R_n = 4.5\times 10^6$ are numerically simulated and the results are compared with those of Sarda's experiment[8] shown in the figures. Fig. 4 shows variations of the pressure coefficient C_P along waterline where measured and computed results show good agreements.

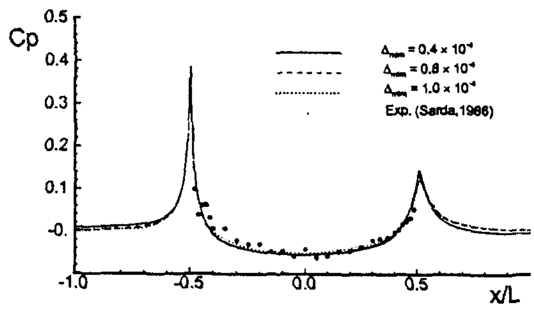


Fig. 4 Variation of Pressure Coefficient along Waterline (Wigley, $R_n = 4.5\times 10^6$

Fig. 5 shows measured and computed iso-wake contours with various grid resolutions at the A.P. section. Indicating that finer grid yield better agreements with measurement and $\Delta_{min} = 2.0\times 10^{-5}$ is sufficient for the case tested.

Fig. 6 compares velocity profiles at the various locations along the waterline. The velocity profiles agree well to the measured data.

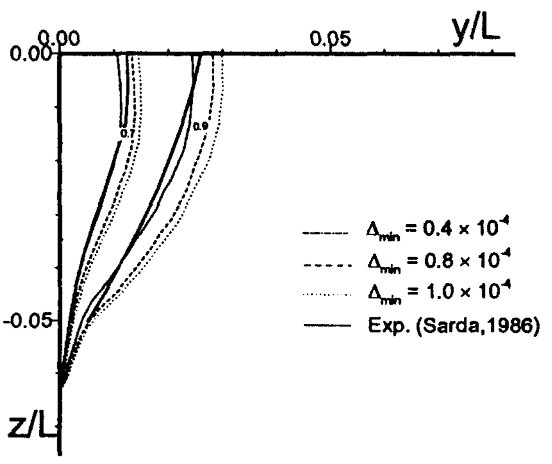


Fig. 5 Iso-wake Contours at the A.P. (Wigley, $R_n = 4.5\times10^6$

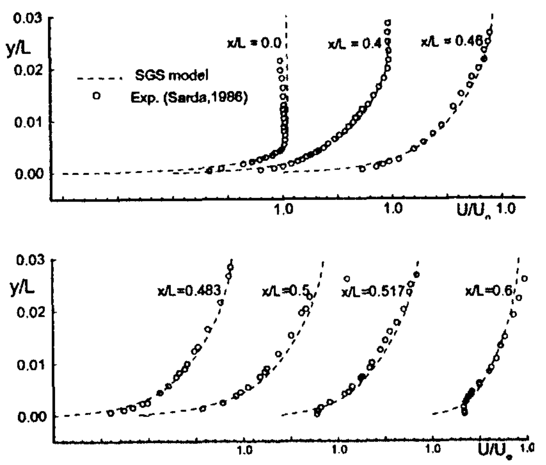


Fig. 6 Comparisons of Velocity Profiles (Wigley, $R_n = 4.5\times10^6$

4.2 Series 60, C_B=0.8

Computational results for the turbulent flow field around the Series 60, C_B=0.8 hull form are shown in the figures. The Reynolds number for computations is chosen to be 2.1×10^6. Fig. 7 shows the comparisons of iso-wake contours corresponding to three different Δ_{min} at the 1/2 station and the propeller plane.

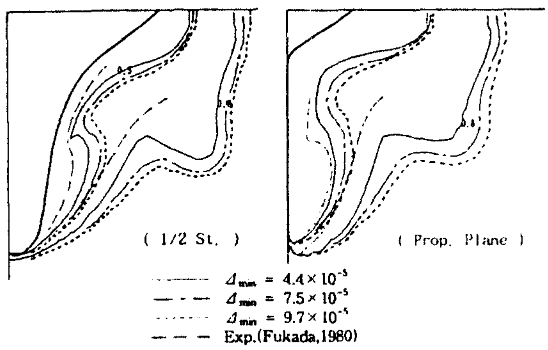
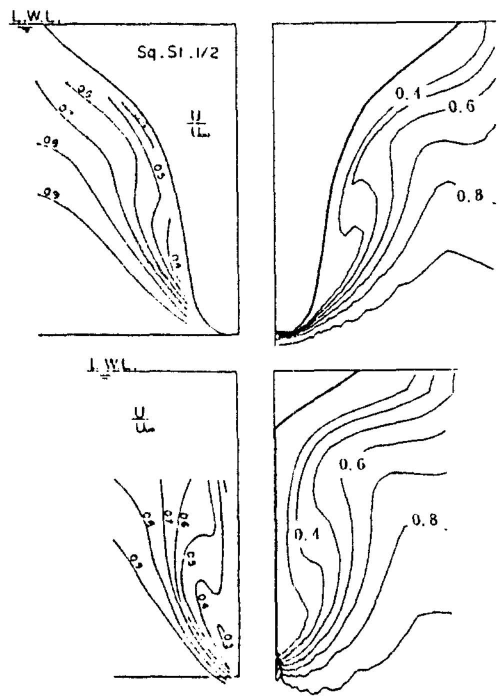


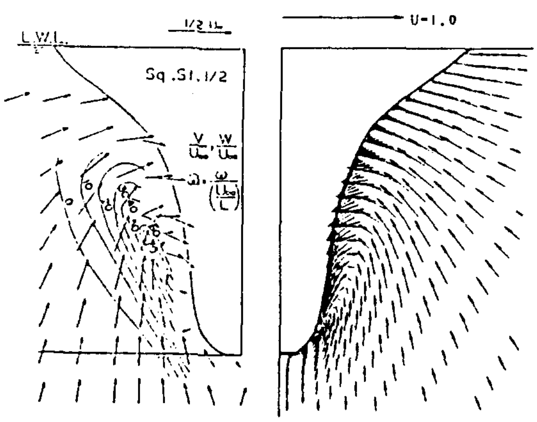
Fig. 7 Comparison of Iso-wake Contours according to Grid Sizes (Series 60, C_B=0.8, $R_n = 2.1\times10^6$)



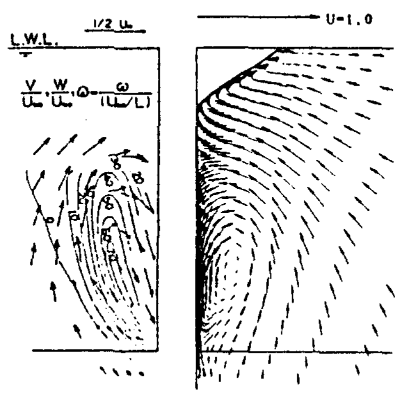
(Prop. Plane)

Fig. 8 Iso-wake Contours (Series 60, $C_B = 0.8$ $R_n = 2.1\times10^6$)

Fig. 8 and Fig. 9 show iso-wake contours and secondary velocity vectors at the identical locations. Comparisons to the experimental results of Fukuda[9] indicates numerical results for finer grid show better agreements to measured ones. Also Fig. 10 shows simulated results for pressure field around the hull.



(1/2 St.)



(Prop. Plane)

Fig.9 Secondary Velocity vectors(Series 60, $C_B = 0.8$ $R_n = 2.1\times10^6$)

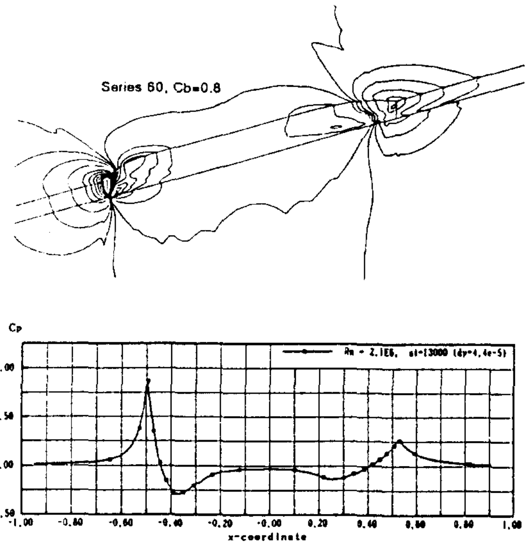


Fig. 10 Pressure Distribution (Series 60, $C_B = 0.8$ $R_n = 2.1\times10^6$

5. CONCLUSIONS

A numerical study on the grid dependency of the numerical solution is performed for a mathematical Wigley hull and the Series 60, $C_B = 0.8$ hull form. The numerical results with various grid sizes were compared with each other and with the measured data and showed favorable trends and agreements. The sub-grid scale turbulence model used in the present study is proven to adequatly recognize the bilge vortex. It is also found that finer grid spacing yield better agreement with experimental data. And $\triangle_{min}$, the minimum grid spacing adjacent to the wall of less than $\frac{0.05}{\sqrt{R_n}}$ is sufficient for the cases tested in the present study.

Further tests and refinements of the present numerical method are necessary, but it can be safely used for predictions of ship stern flow, at least in the qualitative sense.

References

[1] Larsson L., Patel V.C. and Dyne G. ed., "Ship Viscous Flow", Proceedings of 1990 SSPA-CTH-IIHR Workshop, 1991

[2] Baldwin, B. S. and Lomax, H., "Thin layer approximation and algebraic model for separated turbulent flows", AIAA, 16th Aero. Sci. meeting, 1978

[3] Miyata, H., Zhu M. and Watanabe O., "Numerical Study on a Visous Flow with Free-Surface Waves about a Ship in Steady Straight Course by a Finite-Volume Method", JSR, Vol. 36, No. 4, Dec. 1992

[4] Kang, K. J., "Numerical simulation of the viscous flows around ships in a generalized coordinates system", Ph.D Thesis, Inha Univ., 1991

[5] Lee S. H. and Kang K. J., "A numerical computation of the flow field around a Series 60, CB=0.8", The Proceeding of Korea-Japan Joint Workshop on Hydro-dynamics in Ship Design, pp. 233-243., 1991

[6] Koadma, Y., "A Globally Conservative Upwind Scheme Applied to Flow Past a Ship Hull", Proceedings of the Second Osaka International Colloquium on Viscous Fluid Dynamics in Ship and Ocean Technology, 1991

[7] Koadma, Y., "Computation of Ship's Resistance using an NS Solver with Global Conservative - Flat Plate and Series 60, CB=0.6 Hull -", JSNA Japan, Vol. 172, 1992

[8] Sarda, O. P., "Turbulent flow past ship hulls - An experimental and computational study", Ph.D Thesis, Iowa Univ., 1986

[9] Fukuda, K. and Fujii, "A., Turbulence Measurements in Three Dimensional Boundary Later and Wake around a Ship Model", JSNA, Japan, Vol. 150, 1980

[10] Smagorinsky, J., Manebe, S., and Holloway, J. L., "Numerical Results from a nine-level general circulation model of the atmosphere", Monthly Weather Review, 93, Dec., 727-768, 1965

[11] Takakura, Y., Ogawa, S., and Ishiguro, T., "Turbulence Model for Transonic Viscous Flow", AIAA paper, No. 89-1952CP, 1989.

[12] Thompson J.F., Warsi Z.U.A. and Mastin C. W., "Numerical grid generation", North-Holland, 1985

[13] Leonard B.P., "A Stable and accurate convective modeling procedure based quadratic upstream interpolation, Computational Methods", Appl. Mech. Eng. Vol. 19, 1979

[14] Hirt C.W.,Nichols B.D. and Romero N.C., "SOLA-A Numerical solution algorithm for transient fluid flows", Los Alamos Lab., UC-34 and UC-79d, 1979.