Viscous Flow Computations around a Ship Using One-Equation Turbulence Models

by Takanori Hino*, Member

Summary

Two one-equation turbulence models proposed recently were applied to Navier-Stokes computations of viscous flows around ship hulls. The turbulence models consist of a single advection/diffusion equation for eddy viscosity. Flows around two tanker models, the HSVA Tanker and the Dyne Tanker, as well as a flat plate were computed with these turbulence models. Results were compared with measured data and the computations with the conventional algebraic turbulence model. Resistance values computed with the one-equation models were found to be reasonably accurate compared with other numerical results. The wake distributions predicted by the present models showed more rounded contours which were improved from the ones by the algebraic model.

1. Introduction

With the evolution of computer hardware and development of Computational Fluid Dynamics techniques, Navier-Stokes (NS) solutions become more and more commonly used in a viscous flow analysis for ships. Navier-Stokes solvers for ship flows are expected to give an accurate estimation of ship's resistance as well as detailed information of flow field around a ship hull, such as a wake distribution on a propeller plane. Many numerical methods have been proposed for ship flows and efforts are being devoted to development of more accurate and more efficient flow solvers.

Performance of various numerical methods has been compared in several workshops. The CFD Workshop TOKYO 1994 was the latest workshop for ship flows. In the Workshop, the computational results for the specified test cases were gathered and compared with each other. Among the test cases, the double-model flows around the HSVA Tanker hull and the Mystery Tanker hull (named the Dyne Tanker in the Workshop, because the ship was no longer a mystery) were selected as the follow-up of the preceding workshop on ship viscous flows. The results for these cases showed that resistance of full hull forms estimated by the most NS solvers agreed reasonably well with that of model tests. However, the predictions of wake distributions were not so good as those of resistance. Particularly, the so-called "hook" shape in the wake contour which is associated with the longitudinal bilge vortices did not be simulated accurately in the computations with the conventional turbulence models (algebraic models or \( k-\varepsilon \) models). Since the results with Reynolds stress models were much better than the other results presented in the Workshop, it turned out that turbulence models were the key to improve the wake estimation. Although Reynolds stress models were most promising, they require huge computer resources. From the engineering point of view, simpler models which can give a reasonable prediction are favorable.

Recently, new turbulence models which employ one equation have been proposed in the aerodynamics field. These models solve the transport equation for kinematic eddy viscosity. They are expected to be more flexible than algebraic models such as the Baldwin-Lomax and to be simpler than two equation models and much simpler than Reynolds stress models. Applications of the models to flows around a wing with a slat and a flap showed good performance. In reference, it was reported that one-equation models gave better results than \( k-\varepsilon \) models for backward-facing as well as forward-facing step flows.

In the present study, two models, the Spalart-Allmaras model and the \( \nu-92 \) model, both of which are one-equation model, are applied to incompressible viscous flows around an advancing ship. The HSVA Tanker and the Dyne Tanker were selected as test cases. The numerical results are compared with the computation using the Baldwin-Lomax model and the measured data and applicability of one-equation turbulence models to viscous flows around a ship hull is discussed.
2. Navier-Stokes Solver

The flow solver used in the present work is the code called FRESH\(^{(1)}\) which simulates flows around an advancing ship with or without a free surface using Reynolds averaged Navier–Stokes equations. Since the numerics for free surface flows are described in reference\(^{(1)}\), the numerical procedure for double model flows are briefly given below.

The governing equations are Reynolds averaged Navier–Stokes equations for three-dimensional incompressible fluid. With an introduction of artificial compressibility, they can be written as

\[
\frac{\partial \rho}{\partial t} + \rho \frac{\partial v_i}{\partial x_i} = 0
\]

\[
\frac{\partial v_i}{\partial t} + \frac{\partial (\rho v_i v_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} = 0
\]

where

\[
\frac{\tau_i}{\rho} = \left( \frac{1}{R_i} + \frac{1}{R_0} \right) \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right)
\]

\((x_1, x_2, x_3) = (x, y, z)\) are the Cartesian coordinates, \((u_i, v_i, w_i) = (u, v, w)\) are the velocity components and \(p\) is pressure. Variables are nondimensionalized using ship length \(L\), uniform velocity \(U\) and density of water \(\rho\).

The Reynolds number \(R\) is defined as \(UL/\nu\) where \(\nu\) is kinematic viscosity of water. Nondimensional kinematic eddy viscosity \(1/R_i(=\nu_i/UL)\) is determined by a turbulence model. The parameter \(\beta\) represents artificial compressibility and \(\beta = 1\) is used in the present computations.

A finite-volume approach with a cell-centered layout is adopted for spatial discretization. The convective fluxes are evaluated by the third order accurate upwind scheme of MUSCL type based on the flux difference splitting, while the viscous fluxes are evaluated by central differencing scheme.

The boundary conditions are as follows: At the inflow boundary, velocity is specified as \((u, v, w) = (1, 0, 0)\) and pressure condition is \(\partial p/\partial z = 0\), where \(z\) is the streamwise grid direction. At the outflow boundary, \(\partial u_i/\partial x_i = 0\) and \(p = 0\) are given. At the side boundary, \(\partial u_i/\partial x_i = 0\) and \(p = 0\) where \(x\) is the grid direction from a hull to the outer boundary. On the body surface, \((u, v, w) = (0, 0, 0)\) together with \(\partial p/\partial z = 0\) are given as no-slip conditions. The conditions on the center plane and the water plane are \(y\)-symmetry and \(z\)-symmetry.

The present scheme is designed to obtain a steady state solution efficiently at the expense of time accuracy. The explicit five-stage Runge-Kutta scheme is used as the time integration method which is proved to be stable and efficient for steady flow computations. To accelerate convergence of a solution to steady state, multigrid method are employed in the present scheme.

3. Turbulence Models

3.1 Spalart–Allmaras Model

Recently Spalart and Allmaras\(^{(6)}\) proposed a new turbulence model which employs one-equation for eddy viscosity. Starting point of the model is the assumption that kinematic eddy viscosity \(\nu_i\) is governed by a single advection/diffusion equation with source terms. Each terms of the transport equation are calibrated using data of free shear flows and boundary layer flows.

First, kinematic eddy viscosity \(\nu_i\) is defined using the working variable \(\tilde{\nu}\) as,

\[
\nu_i = \nu f_{vi}, \quad f_{vi} = \frac{x_i^T}{x_i^T + c_{vi}^T}, \quad x_i^T = \frac{\tilde{\nu}}{\nu}
\]

where \(\tilde{\nu}\) is molecular viscosity. The motivation for using \(\tilde{\nu}\) rather than \(\nu\) comes from the fact that when \(\nu\) is used very fine grid spacing and a careful numerical treatment are required near a solid wall due to the highly nonlinear behavior of \(\nu\). On the other hand, \(\tilde{\nu}\) behaves linearly near a solid wall and these difficulties do not arise.

The working variable \(\tilde{\nu}\) is assumed to be governed by the following equation,

\[
\frac{\partial \tilde{\nu}}{\partial t} + \nu \frac{\partial \tilde{\nu}}{\partial x_i} =
\]

\[
= c_{\nu}(1 - f_{\nu}) \tilde{\nu} + \frac{1}{\partial x_i} \left( \frac{\partial}{\partial x_i} \left( \nu + \frac{\rho}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right) \right)
\]

\[
+ c_{\nu} \left( \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right)
\]

\[
- \frac{c_{\nu} f_{\nu} - c_{\nu}^T}{\nu} \left( \frac{\partial \tilde{\nu}}{\partial x_i} \right) + f_{\nu}^T \frac{\partial U^2}{\partial x_i} (5)
\]

where the first term of RHS is the production and \(\tilde{S}, f_{\nu}\) and \(f_{\nu}^T\) are

\[
\tilde{S} \equiv S + \frac{\nu}{x_i} \frac{\nu}{x_i} f_{\nu}, \quad f_{\nu} = 1 - \frac{x_i}{x_i + c_{\nu}^T} \quad (6)
\]

\[
f_{\nu}^T = c_{\nu} \exp \left( - c_{\nu} x_i^2 \right) \quad (7)
\]

\(S\) is the vorticity magnitude and \(d\) is the distance from the closest wall. The second and the third terms of RHS are the diffusion and the destruction. The function \(f_{\nu}\) is

\[
f_{\nu} = \frac{1}{\partial x_i} \left( \frac{1 + c_{\nu}^T}{g_i + c_{\nu}^T} \right)^{1/g_i}, \quad g_i = r + c_{\nu} e^{x_i - r}, \quad r = \frac{\tilde{\nu}}{S_k x_i d_i^2} \quad (8)
\]

The fourth term is the trip function. In the Spalart–Allmaras model, transition to turbulence is not modeled and a user must specify the location of transition. The trip function is a source term which is active only near the transition line (called a tripped) and the function \(f_{\nu}^T\) is written as follows:

\[
f_{\nu} = c_{\nu} g_1 \exp \left( - c_{\nu} \frac{a_1 g_1}{d_i^2} (d_i^2 + a_1 d_i^2) \right) \quad (9)
\]

\[
g_1 = \min \left( 0.1, \frac{\Delta U}{\omega_d \Delta x_i} \right) \quad (10)
\]

where \(d_i\) is the distance from the field point to the closest trip on the wall. \(\omega_d\) is the vorticity at the trip, \(\Delta U\) is the difference of velocity between the field point and the trip point and \(\Delta x_i\) is the grid spacing along the wall at the trip.

The model constants are given as follows:
\[ F_2 = \frac{q^2 + 1.3 + 0.2}{q^2 - 1.3 + 0.1}, \quad q = \nu_c / (7
u) \]  

(14)

where \( \nu_c \) is the value of \( \nu_t \) on the wall which is zero for the smoothed wall. The quantities \( I_1, I_2, N_1 \), and \( N_2 \) are defined as

\[ I_1^2 = \frac{\partial \nu_t}{\partial x_j} \left( \frac{\partial \nu_t}{\partial x_j} + \frac{\partial \nu_t}{\partial x_i} \right), \quad I_2^2 = \frac{N_1}{N_2} \left( \frac{\partial \nu_t}{\partial x_i} \right)^2 \]

The model constants are as follows:

- \( A_1 = -0.5 \), \( A_2 = 4.0 \)
- \( C_1 = 0.8 \), \( C_2 = 1.6 \), \( C_3 = 0.1 \), \( C_4 = 4.0 \)
- \( C_5 = 2.9 \), \( C_7 = 31.5 \), \( C_9 = 0.1 \)

It should be noted here that some terms of the original equations associated with compressibility effects are neglected in the expressions above, because all applications in the present study are for incompressible flows.

Since the transition from laminar to turbulent flows is not included in the model, the trip function identical to the one used in the Spalart-Allmaras model is added to the above equation. A numerical procedure for this model is the same as the one for the Spalart-Allmaras model.

3.3 Baldwin-Lomax Model

For comparison, the Baldwin-Lomax model, the widely-used zero equation model, is also employed. The model is based on a two-layer approach, in which kinematic eddy viscosity is given by

\[ \nu_t = \begin{cases} \nu_t^{inner}, & y \leq y_i \\ \nu_t^{outer}, & y > y_i \end{cases} \]

(15)

where \( y_i \) and \( y_c \) are the distance to the wall and the value where \( \nu_t^{inner} = \nu_t^{outer} \), respectively.

In the inner region, \( \nu_t^{inner} \) is given by the Prantl-Van Driest formula as,

\[ \nu_t^{inner} = C_{D_1} \nu_t d_k, \quad C_{D_1} = 0.3 \]

(16)

where \( C_{D_1} \) is a constant. In the outer region, the following expression is used.

\[ \nu_t^{outer} = K \nu_t \]  

(17)

where \( K \) is a constant. \( F_{wake} \) is defined as

\[ F_{wake} = \min \left( \nu_t^{inner} \right) \]

(18)

The model constants are as follows:

- \( A^1 = 26 \), \( C_{D_1} = 0.3 \), \( C_{D_2} = 0.1 \), \( C_{D_3} = 4.0 \)
- \( C_1 = 1.6 \), \( C_2 = 31.5 \), \( C_3 = 0.1 \)

The value of \( C_{D_2} \) is modified from the original value (=0.25) as suggested by Renze et al. 12
4. Results and Discussions

4.1 Flat Plate Flows

First, the present numerical scheme was applied to flows around a flat plate to see behavior of the turbulence models for a simple flow configuration.

Table 1 shows the conditions of computation for a flat plate. The grid used is an orthogonal rectangular mesh generated by an algebraic method. The computed frictional resistance coefficients for three turbulence models are shown in Table 2 and in Fig. 1 together with the empirical value by Schoenherr's formula. The coefficients obtained by the Spalart-Allmaras (SA) model and by the Baldwin-Lomax (BL) model are close to each other and agree well with Schoenherr's value. \( \nu_t - 92 \) (NT) model gives a slightly lower drag than the other two models.

The distributions of friction velocity on a flat plate are shown in Fig. 2. Also plotted in the figure is the empirical curve of Karman-Schoenherr. Except for the region close to the leading edge, three computations make little difference. However, in the leading edge region, friction velocity of the NT model decreases rapidly and then jumps up to the level of the empirical curve. From the examination of \( \nu_t \) distribution in this region, it turns out that the tripping term added to the NT model does not work in this case and that a flow remains laminar. Also, the SA and the BL models show slightly different distributions in the leading edge region. Noted that the grid spacing is relatively coarse to resolve the thin boundary layer of this region in either case.

Figs. 3 show the velocity and kinematic eddy viscosity distributions. Both at \( x = 0.1 \) and at \( x = 0.296 \), the computed velocity distributions with three models agree well with the law of the wall. The kinematic eddy viscosity distributions show good agreement among three models except that the SA model gives higher peak values.

4.2 Ship Flows

The HSVA Tanker and the Dyne Tanker are selected as test cases, because these particular hulls have been the test cases in the past workshops and therefore the extensive experimental data and numerical results are available. The principal dimensions of the ships are presented in Table 3 and the body plans as well as the profiles are shown in Fig. 4. The breadth \( B \), the draft \( d \) and the framelines of the fore part of the two ships are identical, while the aft framelines are different. The Dyne hull is designed to have more U-shaped framelines than the HSVA Tanker. For any turbulence models and numerical schemes, it is very important whether or not they can simulate differences of flow fields for ship

<table>
<thead>
<tr>
<th>Table 1 Conditions of computations for a flat plate</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Model</strong></td>
</tr>
<tr>
<td>Reynolds Number</td>
</tr>
<tr>
<td>Grid</td>
</tr>
<tr>
<td>Minimum Spacing</td>
</tr>
<tr>
<td>Multigrid</td>
</tr>
<tr>
<td>Solution Domain</td>
</tr>
<tr>
<td>Tripping</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2 Resistance coefficients of a flat plate</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Model</strong></td>
</tr>
<tr>
<td>Schoenherr</td>
</tr>
<tr>
<td>Spalart-Allmaras Model</td>
</tr>
<tr>
<td>( \nu_t - 92 ) Model</td>
</tr>
<tr>
<td>Baldwin-Lomax Model</td>
</tr>
</tbody>
</table>

![Fig. 1 Comparison of resistance coefficients of a flat plate.](image)

![Fig. 2 Distributions of friction velocity on flat plate.](image)

![Fig. 3 Distributions of friction velocity on flat plate.](image)

<table>
<thead>
<tr>
<th>Table 3 Principal dimensions of two tanker models</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Model</strong></td>
</tr>
<tr>
<td>( B/L_{pp} )</td>
</tr>
<tr>
<td>( d/L_{pp} )</td>
</tr>
<tr>
<td>( C_B )</td>
</tr>
<tr>
<td>( C_S = S/\sqrt{L_{pp}} )</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3 Principal dimensions of two tanker models</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Model</strong></td>
</tr>
<tr>
<td>( B/L_{pp} )</td>
</tr>
<tr>
<td>( d/L_{pp} )</td>
</tr>
<tr>
<td>( C_B )</td>
</tr>
<tr>
<td>( C_S = S/\sqrt{L_{pp}} )</td>
</tr>
</tbody>
</table>
Viscous Flow Computations around a Ship Using One-Equation Turbulence Models

Fig. 3 Velocity and kinematic eddy viscosity distribution on a flat plate at $x=0.1$ and at $x=0.296$.

Fig. 4 Body plans and profiles of HSVA Tanker and Dyne Tanker.
hulls with small modifications.

The conditions of computation are presented in Table 4. The grids are generated by the algebraic/geometric method. The convergence criterion of flow computations is that $L^2$ norm of the divergence of velocity drops five orders of magnitude from the initial residual. At this stage, the resistance value converges to four or five significant digits.

### 4.2.1 HSVA Tanker

The computed resistance coefficients are shown in Table 5 and in Fig. 5. Included is the experimental value at CETENA, where $C_F$ is obtained from the ITTC 1957 line and $C_T = (1 + K)C_F$ with $K = 0.217$. The total resistance coefficients computed with the SA model and with the NT model are almost identical and both are larger than that with the BL model. The experimental value is in between the two groups. The difference comes mainly from the amounts of the frictional resistance and the pressure drag is virtually independent of turbulence models.

Figs. 6 and Fig. 7 show the comparisons of the hull surface pressure distributions. The overall pressure distributions are similar to each other, which corresponds to little difference of pressure drags. Figs. 7 indicates that the negative peak of pressure at the bilge of the stern is slightly pronounced in the SA and the NT models while measured data shows yet lower value.

$C_F$-distributions along the waterline and the keel line are shown in Fig. 8. Again, three distributions are nearly identical except for the slight difference at the stern region. Agreement with computations and measurement on the keel line are good except for the negative peak at the stern, where the peak is lower in the measurement as seen in the contour plot (Fig. 7).

Figs. 9 show the distributions of friction velocity along the waterline and the keel line. Large discrepancy among three computations can be seen in the stern of the waterline, where the BL model produces the lower friction than the other two models. This is consistent with the lower frictional resistance of the BL model seen in Fig. 5. Compared with Fig. 8, it is found that the difference of friction velocity occurs in the region of adverse pressure gradient. Since the BL model does not include the pressure gradient effect, the one-equation models may work better than the BL model there. In the bow region, behavior of the NT model is different from the other two models, and the flow becomes turbulent from the bow-end in the NT model as well as in the other two models in this case. One of the reasons may be that the tripping term of the NT model is borrowed from the SA model without any modifications. In addition to that, the NT model uses the $\nu_t$ itself as the transport quantity, while the BL model uses the working variable $\tilde{v}$ (See Eq. (4)). As mentioned earlier, the near wall behavior of $\nu_t$ is different from $\tilde{v}$ and the finer grid spacing may be needed for the NT model, particularly in the thin boundary layer at the bow. Further investigations are required for this issue.

Figs. 10 and 11 show the flow fields at $x = 0.447$ and $x = 0.476$. At $x = 0.447$, the velocity distributions of three computations are almost the same and all the computations differ from the measurement with respect to a bulge of the low speed region near the wall. This and the fact that the negative pressure peak at the stern bilge is not simulated well in the computations suggest that the longitudinal bilge vortices in the boundary layer are not captured well by either turbulence model. Kinematic eddy viscosity of the BL model shows the higher maximum value and the wider distribution than the SA and the NT models. The SA and the NT models give similar distribution pattern, though the level of the NT is a little lower than the SA.

At $x = 0.476$, the measured wake contour shows the so-

---

### Table 4 Conditions of computations for tanker models

<table>
<thead>
<tr>
<th>Model</th>
<th>HSVA Tanker and Dyne Tanker</th>
<th>Even Keel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trim</td>
<td>5 x 10^6</td>
<td></td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>113 x 41 x 41</td>
<td></td>
</tr>
<tr>
<td>Grid</td>
<td>(Streamwise, Girth, Normal dir.)</td>
<td></td>
</tr>
<tr>
<td>Minimum Spacing</td>
<td>2.1 x 10^{-3}, 9.5 x 10^{-4}, 4.3 x 10^{-6}</td>
<td></td>
</tr>
<tr>
<td>Multigrid</td>
<td>4 level W-cycle with 3 sequence FMG</td>
<td></td>
</tr>
<tr>
<td>Solution Domain</td>
<td>-2.0 ≤ x ≤ 2.0 ( fp = -0.5, ap = 0.5)</td>
<td></td>
</tr>
<tr>
<td>Tripping</td>
<td>-1.0 ≤ z ≤ 0</td>
<td>Bow End</td>
</tr>
</tbody>
</table>

---

### Table 5 Resistance coefficients of HSVA Tanker

<table>
<thead>
<tr>
<th>Model</th>
<th>$C_F$</th>
<th>$C_P$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment at CETENA</td>
<td>3.40</td>
<td>—</td>
<td>4.11</td>
</tr>
<tr>
<td>Spalart–Allmaras Model</td>
<td>3.71</td>
<td>0.702</td>
<td>4.41</td>
</tr>
<tr>
<td>$\nu_t$–92 Model</td>
<td>3.67</td>
<td>0.715</td>
<td>4.39</td>
</tr>
<tr>
<td>Baldwin–Lomax Model</td>
<td>3.29</td>
<td>0.740</td>
<td>4.03</td>
</tr>
</tbody>
</table>

---

Fig. 5 Comparison of resistance coefficients of HSVA Tanker.
called "hook" shape and the cross flow vectors show the strong longitudinal vortex. The BL model gives a flat wake distribution with large eddy viscosity, while the SA model shows a rounded shape of wake contours and the eddy viscosity level is about one third of that of the BL model. The longitudinal vortex of the SA model is slightly stronger than that of the BL, though the intensity is not so large as in the measurement. The NT result is similar to the SA with more rounded wake contour lines and lower eddy viscosity.

4.2.2 Dyne Tanker

The second series of computations were carried out for the Dyne model with the SA and NT models. The grid properties are kept consistent with the case with the HSVA Tanker. The resistance coefficients are tabulated in Table 6 and depicted in Fig. 12. Though the experimental results are not available, the approximately 3.5% increase of total resistance from the HSVA model is reasonable compared with other numerical results. Resistance increase is due to the pressure drag rather than the frictional drag. Hull surface pressure distributions are shown in Figs. 13. Again, differences between two models are little.

Flow fields at \( x = 0.421 \) and \( x = 0.489 \) are compared in Figs. 14 and 15. In general, the U-shaped framelines cause stronger longitudinal vortices than the V-shaped hull and the wake contour lines become more distorted. These characteristics can also be seen in the present measured data. At \( x = 0.489 \), the measured wake contours make an island which correspond to the stronger longitudinal vortex. The computations reproduce, to some extent, this tendency. The computed wake contours at \( x = 0.489 \) now shows the small "hook", which is not observed in the HSVA case (Figs. 11). Again, the NT model gives slightly better results than the SA model. However, the tendency that the computations do not reproduce well the strong longitudinal vortices and the distorted wake contours remains the same as in
Fig. 7 Hull surface pressure distributions (aft part) on HSVA Tanker, contour interval $\Delta C_p = 0.02$.

Fig. 8 $C_p$ distributions along waterline and keel of HSVA Tanker.

Fig. 9 Distributions of friction velocity along waterline and keel of HSVA Tanker.
Fig. 10 u contour, (v, w) vectors, 1/R_t contour and ω_x contour on x=0.447 of HSVA Tanker.
Fig. 11  $u$ contour, $(u, w)$ vectors, $1/R_t$ contour and $\omega_x$ contour on $x=0.476$ of HSVA Tanker.
Viscous Flow Computations around a Ship Using One-Equation Turbulence Models

4.3 Discussions

For a flat plate flow, frictional resistance of the SA model is almost the same as the BL, while the NT gives lower value because of the different transition position. Except that point, flow field comparisons show that performance of the one-equation models is as good as that of the BL model for a flat plate flow. For ship flows, the resistance coefficients computed by the one-equation models are in the same order of accuracy as those with the conventional turbulence models, though, frictional resistance components of the one-equation models are slightly higher than those of the BL model or the empirical estimation.

With respect to the wake distributions, although the SA and the NT results are improved from the BL, they do not reproduce the "hook" clearly and the intensity of the longitudinal vortices are not so strong as the measurement. The NT model gives slightly better results than the SA model. As mentioned above, the NT model is calibrated using the data of axisymmetric flows in addition to plane flows. On the other hand, the SA model does not include any effects of axisymmetric flows. This difference is partly attributed to better performance of the NT model, since ship stern flows can be approximated by axisymmetric flows more reasonably, though it is still a crude approximation, than by plane flows.

Deng et al. suggested that with the reduction of eddy viscosity with the factor of 2.5 at the core of the longitudinal vortex the "hook" shape can be generated. This suggests that too much eddy viscosity is one of the reasons why the "hook" is not simulated well. One of the weak points of the conventional turbulence models

Table 6 Resistance coefficients of Dyne Tanker

<table>
<thead>
<tr>
<th></th>
<th>$C_F \times 10^3$</th>
<th>$C_P$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart–Allmaras Model</td>
<td>3.76</td>
<td>0.809</td>
<td>4.56</td>
</tr>
<tr>
<td>$\nu_1$-92 Model</td>
<td>3.72</td>
<td>0.825</td>
<td>4.54</td>
</tr>
</tbody>
</table>

Fig. 12 Comparison of resistance coefficients of Dyne Tanker.

Fig. 13 Hull surface pressure distributions on Dyne Tanker, contour interval $\Delta C_p = 0.02$. 

the HSVA case.

Table 6 Resistance coefficients of Dyne Tanker

<table>
<thead>
<tr>
<th></th>
<th>$C_F \times 10^3$</th>
<th>$C_P$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart–Allmaras Model</td>
<td>3.76</td>
<td>0.809</td>
<td>4.56</td>
</tr>
<tr>
<td>$\nu_1$-92 Model</td>
<td>3.72</td>
<td>0.825</td>
<td>4.54</td>
</tr>
</tbody>
</table>

Fig. 12 Comparison of resistance coefficients of Dyne Tanker.

Spalart–Allmaras Model

$\nu_1$-92 Model

Fig. 13 Hull surface pressure distributions on Dyne Tanker, contour interval $\Delta C_p = 0.02$. 

the longitudinal vortices are not so strong as the measurement. The NT model gives slightly better results than the SA model. As mentioned above, the NT model is calibrated using the data of axisymmetric flows in addition to plane flows. On the other hand, the SA model does not include any effects of axisymmetric flows. This difference is partly attributed to better performance of the NT model, since ship stern flows can be approximated by axisymmetric flows more reasonably, though it is still a crude approximation, than by plane flows.

Deng et al. showed that with the reduction of eddy viscosity with the factor of 2.5 at the core of the longitudinal vortex the "hook" shape can be generated. This suggests that too much eddy viscosity is one of the reasons why the "hook" is not simulated well. One of the weak points of the conventional turbulence models

Table 6 Resistance coefficients of Dyne Tanker

<table>
<thead>
<tr>
<th></th>
<th>$C_F \times 10^3$</th>
<th>$C_P$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart–Allmaras Model</td>
<td>3.76</td>
<td>0.809</td>
<td>4.56</td>
</tr>
<tr>
<td>$\nu_1$-92 Model</td>
<td>3.72</td>
<td>0.825</td>
<td>4.54</td>
</tr>
</tbody>
</table>

Fig. 12 Comparison of resistance coefficients of Dyne Tanker.

Spalart–Allmaras Model

$\nu_1$-92 Model

Fig. 13 Hull surface pressure distributions on Dyne Tanker, contour interval $\Delta C_p = 0.02$. 

the longitudinal vortices are not so strong as the measurement. The NT model gives slightly better results than the SA model. As mentioned above, the NT model is calibrated using the data of axisymmetric flows in addition to plane flows. On the other hand, the SA model does not include any effects of axisymmetric flows. This difference is partly attributed to better performance of the NT model, since ship stern flows can be approximated by axisymmetric flows more reasonably, though it is still a crude approximation, than by plane flows.

Deng et al. showed that with the reduction of eddy viscosity with the factor of 2.5 at the core of the longitudinal vortex the "hook" shape can be generated. This suggests that too much eddy viscosity is one of the reasons why the "hook" is not simulated well. One of the weak points of the conventional turbulence models
is, therefore, that they produce excessive amounts of eddy viscosity in a region where longitudinal vortices are strong. In the present results with one-equation models, the eddy viscosity level is lower than that of the conventional zero equation model. At this point performance of the one-equation models is promising enough that the models can be considered as the candidate of the base model of the further tuning. Obviously low eddy viscosity level itself is not sufficient to capture the complex structure of ship stern flows.

The present one-equation models are for two-dimensional, either plane or axisymmetric, flows. Ship stern flows are three dimensional and much more complex than model flows used for calibration of the turbulence models. More precise tuning for complex flows should be made. To achieve this, databases of flow measurements around ship sterns have to be established, where not only mean-flow data but also turbulence quantities should be collected.

5. Conclusions

Two one-equation turbulence models, the Spalart-Allmaras model and the $\nu_t$-92 model, both of which solve the transport equation for eddy viscosity were applied to viscous flow computations around a ship. The computational results show that the resistance evaluation is in the same order of accuracy as that of the conventional turbulence models such as the Baldwin-Lomax model. By comparisons of the flow fields, it turns out that the one-equation models give wake distribution improved from the algebraic model, though the so-called "hook" shape is not reproduced with

Fig. 14 $u$ contour, ($u$, $w$) vectors, $1/R_t$ contour and $\omega_x$ contour on $x=0.421$ of Dyne Tanker.
sufficient accuracy. The level of eddy viscosity in a wake is lower than the conventional models and the models are adequate as a candidate of the base model for further tuning. Performance of the Spalart-Allmaras model and that of the \( \nu_t - 92 \) model are nearly equal. The \( \nu_t - 92 \) model produces better wake distributions, while the Spalart-Allmaras model is simpler and gives reasonable frictional drag. Further assessment with appropriate tuning is required to give the definite preference.

Acknowledgments

The author would like to thank Dr. Y. Kodama of Ship Research Institute for his valuable suggestions and for providing the surface grid generation program. The author would also like to acknowledge the valuable discussions with Mr. N. Hirata of Ship Research Institute regarding numerics of the Spalart-Allmaras turbulence model. Finally, the supports and discussions of CFD group at Ship Research Institute are gratefully acknowledged.

References


