

平成31年4月 田原裕介先生を偲ぶ会 事務局 【ご注意】

この論文集は、あくまでも個人使用目的で作成されたものです。従って、<u>この追悼論文集を</u> 直接、参考文献等として引用することは出来ませんのでご留意下さい。

OBITUARY

Dr. Yusuke Tahara

Dr. Yusuke Tahara, the senior research engineer of National Maritime Research Institute (NMRI), sadly passed away on January 25th 2019 at the age of 56. Dr. Tahara earned his Bachelor of Engineering and Master of Engineering in 1985 and 1987, respectively, both from Yokohama National University. After one-year experience as a system engineer at Japan IBM, he started his postgraduate study in the University of Iowa, IIHR-Hydroscience & Engineering- in 1988. In 1992, he earned his Ph.D. degree with the dissertation title "Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number". Dr. Tahara started his academic carrier as the post-doctoral scholar at IIHR for 2 years, and then returned back to Japan as the assistant professor at Osaka Prefecture University (OPU) in 1994. On 2009, he chose NMRI as the next base of his research carrier after 15 years of activities in OPU, lastly appointed as the associate professor.

Dr. Tahara was the professional in developing codes in computational ship hydrodynamics including potential and viscous flow solvers, grid generation, overset grid assembler and optimization. In the meantime, he put great importance on industrial application of his research works. He was also the yacht scientist. As the core representative of technical team of Nippon Challenge America's Cup during 1995 to 2000, the appendages for two challenger boats, "Asyura" (JPN44) and "Idaten" (JPN52), were designed and equipped owing to enormous efforts of Dr. Tahara and his team. As the research engineer, professor and yacht scientist, the achievement of Simulation Based Design throughout R+3D, e.g. Research, Development, Demonstration and Dissemination, was his life work.

Dr. Tahara was nominated as the member of Resistance Committee in 25th and 26th ITTC. He accomplished many tasks including worldwide round robin tests in towing tanks for establishing benchmark data to identify the facility bias. As the conference operational director, several international workshops were able to be carried out with success, such as Osaka Colloquium, New S-Tech, and AMEC. His contributions to the field of naval architecture and ocean engineering were domestically and internationally awarded, e.g.

SNAME ABS Captain Joseph H. Linnard Prize on 2010, JASNAOE Paper Award on 2012 and so on. Dr. Tahara supervised more than 30 students in OPU, and kept training young researchers after he moved to NMRI. "Face up. Find the way forward. No matter what. We can do it." We were always encouraged by Dr. Tahara's super positive attitude.









掲載論文目次

1.	池畑 光尚,田原 裕介,1987, "船体まわりの自由表面流に及ぼす境界層と伴流の影響",日 本造船学会論文集 第161号 pp.49-57.
2.	Yusuke Tahara, 1992, "A Boundary-Element Method for Calculating Free-Surface Flows Around a Yawed Ship",関西造船協会誌 第218号 pp. 55-67.
3.	Yusuke Tahara, Frederick Stern, B. Rosen, 1992, "An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number", Journal of Computational Physics Vol98, No. 1, pp. 33-53.
4.	Yusuke Tahara, Allen T. Chwang, 1993, "Nonlinear Free – Surface Flow in Oscillating Channel during Earthquakes", Journal of Engineering Mechanics Vol. 119, No. 4, pp. 801-812.
5.	Yusuke Tahara, 1993, "Computation of Viscous Flow around Series 60 Model and Comparison with Experiments",関西造船協会誌 第 220 号 pp. 29-47.
6.	Yusuke Tahara, Frederick Stern, 1994, "Validation of an Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number", Computers&Fluids Vol. 23, No. 6, pp. 785-816.
7.	Yusuke Tahara, 1995, "An Application of Two-Layer k-ɛ Model to Ship Flow Computation", 日本造船学会論文集 第177号 pp. 161-176.
8.	Yusuke Tahara, 1995, "Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht : For a Canoe Body Case", 関西造船協会誌 第 224 号 pp. 1-11.
9.	Yusuke Tahara, Yoji Himeno, 1996, "Applications of Isotropic and Anisotropic Turbulence Models to Ship Flow Computation", 関西造船協会誌 第 225 号 pp. 75-91.
10.	Yusuke Tahara, Satoshi Mitarai, Yoji Himeno, 1996, "A Computational Study of Three-Dimensional Laminar Separation on Prolate Spheroid at Incidence",関西造船協会誌 第 225 号 pp. 93-105.
11.	Yusuke Tahara, 1996, "A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht",関西造船協会誌 第 226 号 pp. 63-76.
12.	Yusuke Tahara, Frederick Stern, 1996, "A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes and Wave Fields for Nonzero Froude Number", Journal of Computational Physics Vol27, pp. 398-411.

13. 田原 裕介, 岩崎 泰典, 1997, "2 次元モデルによるトランサム船尾近傍の自由表面流場に関 する研究", 関西造船協会誌 第227号 pp. 7-19. 14. 田原 裕介, 1997, 1998, "非 H 型トポロジー自由表面パネルを用いた Drawson-Type Rankine-Source 法による定常造波問題の数値計算-第一報[~]第二報", 関西造船協会誌 第 228 号, 第 230 号. 15. 田原 裕介, 姫野 洋司, 1998, "CFD による二次元翼形状改良問題に関する研究", 関西造船 協会誌 第 229 号 pp. 27-35. 16. 沖本 憲司, 姫野 洋司, 田原 裕介, 1998, "時間平均速度勾配テンソルの第2不変量を用い た k-ε 乱流モデルの再構成に関する研究", 関西造船協会誌 第230号 pp. 89-99. 17. 西川 達雄,田原 裕介,正岡 孝治,姫野 洋司,1998, "セールを対象とした粘性流体中に おける空力弾性問題", 関西造船協会誌 第230号 pp. 81-88. 18. 田原 裕介,吉田 豊,西田 隆司, 1998, "RaNS 法によるアメリカ杯レース艇のストラット/ バルブ近傍流場の数値計算および実験との比較",関西造船協会誌 第230号 pp. 141-146. 20. 田原 裕介, 齋藤 泰夫, 姫野 洋司, 松山 博志, 西田 隆司, 安東 潤, 1999-2000, "CFD によるタンカー船型の船尾形状最適化-第1報~第3報", 関西造船協会誌 第231号, 第232 号, 第234号. 21. Yusuke Tahara, 1999, "Wave Influences on Viscous Flow around a Ship in Steady Yaw Motion", 日本造船学会論文集 第186 号, pp. 157-168. 22. 田原 裕介,吉田 豊,西田 隆司,松井 享介,上野 一郎,2000, "アメリカ杯レース艇の バラストバルブ開発-特に基本抵抗形状の検討について-", 関西造船協会誌 第 234 号 pp. 51-59. 23. Yusuke Tahara, Joseph Longo, Frederick Stern, 2002, "Comparison of CFD and EFD for the Series 60 in Steady Drift Motion", Journal of Marine Science and Technology Vol. 7, Issue 1, pp. 17-30. 24. Yusuke Tahara, Tokihiro Katsui, Yoji Himeno, 2002, "Computation of ship Viscous Flow at Full Scale Reynolds Number -Consideration of Near-Wall Flow Model Including Surface Roughness Effects-",日本造船学会論文集 第192号 pp. 89-101. 25. 勝井 辰博, 姫野 洋司, 田原 裕介, 2003, "平板摩擦抵抗値の再検討について", 関西造船 協会論文集 第239号 pp. 37-44. 26. Yusuke Tahara, So Sugimoto, Shinya Murayama, Tokihiro Katsui, 2003, "Development of CAD/CFD/Optimizer-Integrated Hull-Form Design System",関西造船協会論文集 第 240 号 pp. 29-36.

- 27. 田原 裕介,林 豪,2003, "マルチブロック NS/RaNS 法によるアメリカ杯レース艇用風下帆 走セールシステム周りの流場解析",日本造船学会論文集 第194号 pp. 1-12.
- 28. 田原 裕介,勝井 辰博,川崎 正人,児玉 欣二,姫野 洋司,2003, "PC クラスター並列計 算における大規模・高効率 CFD のコーディング技術の開発-第1報:MPI プロトコルを適用した コード開発環境の構築と初期評価-",関西造船協会論文集 第241号 pp. 47-58.

 Yusuke Tahara, Satoshi Tohyama, Tokihiro Katsui, 2006, "CFD-based Multi-objective Optimization Method for Ship Design", International Journal for Numerical Methods in Fluids Vol. 52 pp. 499-527.

30. Yusuke Tahara, Robert V. Wilson, Pablo M. Carrica, Frederick Stern, 2006, "RANS Simulation of a Container Ship using a Single-Phase Level-Set Method with Overset Grids and the Prognosis for Extension to a Self-Propulsion Simulator", Journal of Marine Science and Technology Vol. 11, Issue 4, pp. 209-228.

31. Yusuke Tahara, Daniele Peri, Emillio F. Campana, Frederick Stern, 2008, "Computational Fluid Dynamics-based Multiobjective Optimaization of a surface combatant using a global optimization method", Journal of Marine Science and Technology, Vol.13, Issue 2, pp. 95-116.

- Yusuke Tahara, Daniele Peri, Emilio F. Campana, Frederick Stern, 2011, "Single-and Multiobjective Design Optimization of a Fast Multihull Ship: Numerical and Experimental Results", Journal of Marine Science and Technology, Vol.16, Issue 4, pp. 412-433.
- 33. Kandasamy Manivannan, Peri Daniele, Yusuke Tahara, Wilson Wesley, Miozzi Massimo, Georgiev Svetlozar, Milanov Evgeni, Campana Emilio F., Stern, Frederick, 2013, "Simulation Based Design Optimization of Waterjet Propelled Delft Catamaran", International Shipbuilding

Progress Vol60, pp. 277-308.

34. Yusuke Tahara, Yutaka Masuyama, Toichi Fukasawa, Masanobu Katori, 2014, "Sail Performance Analysis of Sailing Yachts by Numerical Calculations and Experiments", Fluid Dynamics, Computational Modeling and Applications (ISBN: 978-953-51-0052-2).

35. Yusuke Tahara, Yasuo Ichinose, Azumi Kaneko, Yoshikazu Kasahara, 2018, "Variable Decomposition Approach applied to Multi-objective Optimization for Minimum Powering of Commercial Ships", Journal of Marine Science and Technology, Vol.24, Issue 1, pp. 260-283.

船体まわりの自由表面流に及ぼす境界層 と伴流の影響

正員池畑光尚*田原裕介**

Influence of Boundary Layer and Wake on Free Surface Flow around a Ship Model

by Mitsuhisa Ikehata, Member Yusuke Tahara

Summary

The Rankine-source method has been combined with an integral method for boundary layer and wake so as to investigate the influence of boundary layer and wake on the free surface flow around a ship model. The head loss due to viscosity has been taken into account in the equation of the boundary condition on the free surface. The numerical solution of the free surface flow with the boundary layer and wake around 6 m long Wigley model has been obtained with aid of the technique of matrix calculation for source densities on the panel array on the hull surface along with center plane in wake and on the free surface. The computed results as for pressure distributions on the hull surface, wave profiles along the hull side and pressure, frictional and total resistances have been shown in comparison with measurements. They display good results by considering the existence of boundary layer and wake around the hull of the model.

1 緒 言

船体に働く抵抗は、非粘性成分としての造波抵抗、お よび流体の粘性に起因する粘性抵抗に大別される。これ らのうち造波抵抗はポテンシャル理論で取扱えるために 理論的な研究が進み、比較的痩せた船型や低速で航行す る船に関しては、かなりの精度で実験値と合うまでにな った。一方後者の粘性抵抗は、船体表面に働く摩擦抵 抗、境界層の排除影響に基づく圧力抵抗などの抵抗成分 を含んでおり、これらの相互影響も複雑である。粘性抵 抗の算定に貢献してきたのは三次元境界層理論の発達で ある。歴史的には波や造波抵抗の理論よりもかなり遅れ て発達したが、その背景には電子計算機の大型化、高速 化といった計算機技術の飛躍的進歩がある。

ところで、こうして別々に議論されてきた二つの抵抗 成分の間には複雑な相互干渉が存在する。この相互干渉 は、より現実の現象に近い、そしてより実用的な計算を 行うために、ぜひとも考慮されねばならないことである。

粘性影響(境界層影響)と波との相互干渉問題はこれまでにもかなりの研究者によって扱われてきたが、その研

究は大きく二つの観点に分類できる。その一つは造波抵 抗に及ぼす粘性影響を考慮したものである。Havelock¹⁾ は粘性による波の滅衰を表現する reduction factor を 仮定し, thin ship theory を用いて計算した。乾²⁾は船 尾波の粘性滅衰,および船体の存在による閉塞条件によ る修正を提案し, Milgram³⁾は造波抵抗に粘性の与える 影響を考慮した伴流モデルを示した。また,岡部・神 中⁴⁾は境界層の排除厚を船体に付加することにより粘性 影響を考慮した。日夏・足達⁵⁾は境界層影響を考慮した Guilloton 法を用いているが,日夏・竹子⁶⁾では高次の境 界層方程式を用いて,さらに厳密な計算を行っている。 Larsson and Chang⁷⁾は Neumann-Kelvin 近似と高次 境界層理論を用いた。残るもう一つの観点は,自由表面 条件式を導く過程で粘性影響を考慮するものであるが, 姫野・高木⁶⁾,丸尾・松村⁹⁾,茂里¹⁰⁾らが試みている。

本論文においては後者の視点にたち,茂里¹¹⁾が提案し た線形自由表面条件式を参考にし,荻原¹²⁾らによりその 有用性が立証された Rankine-source 法を用い,粘性影 響を考慮した自由表面計算を行うプログラムを開発し た。境界層の計算には積分的解法を用い,伴流まで連続 した計算が可能な池畑ら¹³⁾が提案した反復法を採用す る。荻原¹²⁾らの Rankine-source 法は,二重模型流れを 基底とした波動ポテンシャルについて,線形自由表面条

^{*} 横浜国立大学工学部

^{**} 日本 IBM(株)(研究当時横浜国立大学大学院工学 研究科)

件,および船体表面条件を満足させるものである。これ は Dawson¹⁴⁾により初めて提案された境界要素法を用 いた方法を基に,茂里¹¹⁾の Rankine-source 法の考え方 を参考にして,計算法に改良を加えたものである。さら に荻原¹⁵⁾では非線形影響を考慮した計算も行い,実験 ともよく合う結果を得たが,粘性影響までは考慮してい ない。

粘性影響と波の相互干渉問題を考えるとき,いかなる 要素を用いてお互いの影響を考慮させるかが重要なポイ ントとなる。このとき,その要素を仲立ちとした反復的 解法は非常に有効である。本論文においては,そのよう な近似解法の第一段階として,境界層が波に与える影響 までを考えるために,粘性による Head loss を考慮し た線形自由表面条件式を, 荻原¹²⁾の提案した逐次近似解 法を応用して解いている。

2 粘性影響を考慮した自由表面流の計算法

2.1 境界条件式

まず座標系を Fig.1に示すように定義し,船体に固定 された座標を (x, y, z) とする。ここで,自由表面を平 面壁と見なし,鏡像を考えた二重模型近似の場合の,粘 性を含む船体周囲流場の流速ベクトルを \bar{q}_0 とおく。 Φ_w を自由表面の波動影響を表わす波動ポテンシャルとすれ ば,自由表面波動を考慮した船体周囲流場の流速ベクト ル \bar{q} は次のように表わされる。

$$\vec{q} = \nabla \Phi_w + \vec{q}_0 \tag{1}$$

自由表面形状を

$$z = \zeta(x, y) \tag{2}$$

とするとき,自由表面上で満足されるべき運動学的条件 は

$$\vec{q} \cdot \nabla(z - \zeta) = 0 \quad (z = \zeta) \tag{3}$$

また、圧力の条件は

$$g\zeta + \frac{1}{2} (|\nabla \Phi_w + \vec{q}_0|^2 - U_{\omega}^2) + g\delta H = 0 \quad (z = \zeta)$$
(4)

ここで $g\delta H$ は粘性による Head loss を表わし、次の式 で与えられる。

$$g\delta H = \frac{U_{\infty}^{2}}{2} - \left\{ \frac{|\vec{q}_{0}|^{2}}{2} + \frac{P}{\rho} \right\}$$
(5)



Fig. 1 Coordinate system

したがって、 \hat{q}_0 がポテンシャル流を表わすとき $g\delta H$ は 零となる。 \hat{q}_0 の x 軸方向、y 軸方向、z 軸方向の流速 成分を u_0, v_0, w_0 とし、z=0 で $w_0=0$ とすれば、(3) (4)式は、

$$\begin{aligned} (\varPhi_{wx} + u_0) \zeta_x + (\varPhi_{wy} + v_0) \zeta_y - \varPhi_{wz} = 0 \quad (z=0) \\ & (6) \\ g\zeta + \frac{1}{2} \left\{ (\varPhi_{wx} + u_0)^2 + (\varPhi_{wy} + v_0)^2 + \varPhi_{wz}^2 - U_{\infty}^2 \right\} \end{aligned}$$

(7)

$$+g\delta H=0$$
 (z=0)

(6)(7)よりくを消去すれば

$$\frac{1}{2} (\varPhi_{wx} + u_0) [(\varPhi_{wx} + u_0)^2 + (\varPhi_{wy} + v_0)^2 + \varPhi_{wz}^2 + 2 g \delta H]_x + \frac{1}{2} (\varPhi_{wy} + v_0) [(\varPhi_{wx} + u_0)^2 + (\varPhi_{wy} + v_0)^2 + \varPhi_{wz}^2 + 2 g \delta H]_y + g \varPhi_{wz} = 0 \quad (z = 0) \qquad (8)$$

ここで, z=0における流れに沿う座標を l として, 座標 変換を行う時,以下の関係がある。

 $u_0F_x+v_0F_y=q_0F_l$, ただし $q_0=|\vec{q}_0|$ (9) (8) 式において Φ_w の二乗の項を省略し, かつ上の関 係を適用すれば, Φ_w に関して線形な次の自由表面条件 式を得る。

$$\begin{split} q_{J}(q_{0}\varPhi_{wl})_{l} + \varPhi_{wx}(q_{0}q_{0x} + g\delta H_{x}) \\ &+ \varPhi_{wy}(q_{0}q_{0y} + g\delta H_{y}) + q_{0}g\delta H_{l} + g\varPhi_{wz} \\ &= -q_{0}^{2}q_{0l} \quad (z=0) \quad (10) \\ \text{こで } q_{0} \quad \vec{n} \vec{n} \vec{\tau} \vec{\tau} \vec{\nu} \vec{\nu} + \nu \ddot{n} \vec{\tau} \vec{\sigma} \vec{\delta} \vec{\delta} \vec{\delta}, \quad \vec{L} \vec{\tau} \vec{\zeta} \vec{U} \vec{\Gamma} \vec{O} \end{split}$$

ここで q_0 がポテンシャル流である場合,上式は以下の ようになる。

$$q_0^2 \varphi_{wll} + 2 q_0 q_{0l} \varphi_{wl} + g \varphi_{wz} = -q_0^2 q_{0l} \quad (z=0)$$
(11)

このとき

)

$$q_0 = \Phi_{0l} \quad q_{0l} = \Phi_{0l}$$

なる Φ₀ があるとすれば(11)式は

$$\Phi_{0l}^{2} \Phi_{wll} + 2 \Phi_{0l} \Phi_{0ll} \Phi_{wl} + g \Phi_{wz}$$

$$= - \Phi_{0l}^{2} \Phi_{0ll} \quad (z=0)$$

$$(12)$$

となり、これは Dawson により導かれた線形自由表面 条件式と一致する。

一方,船体表面で $\boldsymbol{\phi}_w$ が満たすべき境界条件は,船体 表面の外向き法線を \boldsymbol{n} とすれば

$$\frac{\partial \Phi_w}{\partial n} = 0 \quad (\text{on } A_0) \tag{13}$$

$$\Phi_w = -\iint_{A_1} \frac{\sigma_1}{r_1} dS_1 - \iint_{A_0} \frac{\sigma_0}{r_0} dS_0 \quad (14)$$

 A_0 は船体表面, dS_0 はその面素, A_1 は自由表面, dS_1 はその面素を表わす。 r_0 , r_1 はそれぞれの面素の位置から流場内の点までの距離を表わす。

(14) 式の右辺第一項は水面の波動ポテンシャルを表わし, 第二項は船体表面に及ぼす波の影響になる。

2.2 数値計算のための離散化

ここでは荻原の方法にならい,方程式の離散化を試みる。

まず船体表面を *M*₀ 個,船体を囲む静水面の有限な領 域を *M*₁ 個の微小面素に分割する。各面素内の吹き出し 強さを一定として (14) 式を用い、*Φ*_{wl} を離散化すれば 以下のようになる。

$$\Phi_{wl}(i) = \sum_{j=1}^{M_1} \sigma_1(j) L_1(ij) + \sum_{j=1}^{M_0} \sigma_0(j) L_0(ij) \\ L_0(ij) = \iint_{\mathcal{J}A_0^j} \frac{\{(x_i - x')u_0 + (y_i - y')v_0\}}{\sqrt{u_0^2 + v_0^2 \cdot r_0^3}} dS_0 \\ L_1(ij) = \iint_{\mathcal{J}A_1^j} \frac{\{(x_i - x')u_0 + (y_i - y')v_0\}}{\sqrt{u_0^2 + v_0^2 \cdot r_1^3}} dS_1$$

$$(15)$$

波の放射条件は ϕ_{wll} を上流差分で計算すれば満足されることが Dawson により示されている。本論文でもそれにならい、 ϕ_{wll} を離散化すれば以下のようになる。

$$\Phi_{wll}(i) = \sum_{j=1}^{M_1} \sigma_1(j) CL_1(ij) + \sum_{j=1}^{M_0} \sigma_0(j) CL_0(ij)$$

$$CL_0(ij) = \sum_{n=0}^{N-1} e_n L_0(i-n,j)$$

$$CL_1(ij) = \sum_{n=0}^{N-1} e_n L_1(i-n,j)$$
(16)

*e_n*は *N* 点上流差分演算子で以下のように定義される。
 When *N*=2

$$F_{il} = e_0 F_i + e_1 F_{i-1}$$
$$e_1 = -e_0 = \frac{1}{(l_{i-1} - l_i)}$$

When N=3

$$\begin{split} F_{il} &= e_0 F_i + e_1 F_{i-1} + e_2 F_{i-2} \\ e_2 &= -\frac{(l_{i-1} - l_i)^2}{(l_{i-1} - l_i)(l_{i-2} - l_i)(l_{i-2} - l_{i-1})} \\ e_1 &= \frac{(l_{i-2} - l_i)^2}{(l_{i-1} - l_i)(l_{i-2} - l_i)(l_{i-2} - l_{i-1})} \\ e_0 &= -(e_1 + e_2) \end{split}$$

When N = 4

 $\varPhi_{wx}(i)\left\{q_{\mathfrak{o}}(i)q_{\mathfrak{o}x}(i)+g\delta H_{x}(i)\right\}$

$$+ \Phi_{wy}(i) \{ q_0(i) q_{0y}(i) + g \delta H_y(i) \} \\= \sum_{j=1}^{M_1} \sigma_1(j) H_1(ij) + \sum_{j=1}^{M_0} \sigma_0(j) H_0(ij)$$
(17)

ただし

$$\begin{split} H_{0}(ij) &= \{q_{0}(i)q_{0x}(i) + g\delta H_{x}(i)\} \iint_{\mathcal{J}A_{0}j} \frac{(x_{i}-x')}{r_{0}^{3}} dS_{0} \\ &+ \{q_{0}(i)q_{0y}(i) + g\delta H_{y}(i)\} \iint_{\mathcal{J}A_{0}j} \frac{(y_{i}-y')}{r_{0}^{3}} dS_{0} \\ H_{1}(ij) &= \{q_{0}(i)q_{0x}(i) + g\delta H_{x}(i)\} \iint_{\mathcal{J}A_{1}j} \frac{(x_{i}-x')}{r_{1}^{3}} dS_{1} \\ &+ \{q_{0}(i)q_{0y}(i) + g\delta H_{y}(i)\} \iint_{\mathcal{J}A_{1}j} \frac{(y_{i}-y')}{r_{1}^{3}} dS_{1} \\ &= t \in \Phi_{wz} \text{ if } U \subset I t \end{split}$$

 $\Phi_{wz} = \begin{cases}
-2\pi\sigma_1(i) & (j=i) \\
0 & (j\neq i)
\end{cases} \text{ on } \Delta A_1^j \quad (18)$

さて,ここで(15)(16)(17)(18)式を(10)式に代入す ることにより,自由表面の境界条件に基づく次の連立方 程式を得る。

$$\begin{split} \sum_{j=1}^{M_{1}} \sigma_{1}(j) A_{1}(ij) + \sum_{j=1}^{M_{0}} \sigma_{0}(j) A_{0}(ij) - 2\pi g \sigma_{1}(i) = B(i) \\ A_{0}(ij) = q_{0}^{2}(i) C L_{0}(ij) + q_{0}(i) q_{0l}(i) L_{0}(ij) + H_{0}(ij) \\ A_{1}(ij) = q_{0}^{2}(i) C L_{1}(ij) + q_{0}(i) q_{0l}(i) L_{1}(ij) + H_{1}(ij) \\ B(i) = -q_{0}^{2}(i) q_{0l}(i) - q_{0}(i) g \delta H_{l}(i) \\ (i=1,2,\cdots M_{1} \text{ on } A_{1}) \\ (19) \end{split}$$

一方,船体表面の境界条件に基づく連立方程式は,(13) 式に(14)式を代入することにより求められ,以下のよ うになる。

$$\begin{bmatrix}
\sum_{j=1}^{M_{1}} \sigma_{1}(j) N_{1}(ij) + \sum_{j=1}^{M_{0}} \sigma_{0} N_{0}(ij) = 0 \\
N_{0}(ij) = -\iint_{JA_{0}^{j}} \frac{\partial}{\partial n} \left(\frac{1}{r_{0}}\right) dS_{0} \\
N_{1}(ij) = -\iint_{JA_{1}^{j}} \frac{\partial}{\partial n} \left(\frac{1}{r_{1}}\right) dS_{1} \\
(i=1, 2, \cdots, M_{0} \text{ on } A_{0})
\end{bmatrix}$$
(20)

2.3 近似計算法の導入

(19) 式と (20) 式は σ_1 , σ_0 についての full matrix の 連立方程式であり、消去法を用いれば解くことが でき る。しかしそのためには $(M_0+M_1)^2$ 規模の正方マトリ クスが必要であり、これを一度に解くのも非効率的であ る。そこでここでも荻原の方法を参照し、近似計算法を 導入する。その過程を以下に箇条書にして示す。

として(19)式に代入すれば、 σ_1 の第一近似解 $\sigma_1^{(1)}$ を 求める連立方程式は以下のようになる。

$$\sum_{j=1}^{M_1} \sigma_1^{(1)}(j) A_1(ij) - 2\pi g \sigma_1^{(1)} = B(i) \qquad (21)$$
$$(i=1,2,\cdots,M_1 \text{ on } A_1)$$

-3-

これを解いて求められた o₁⁽¹⁾ は船体表面上に

$$v_n(i) = \sum_{j=1}^{M_1} \sigma_1^{(1)}(j) N_1(ij) \qquad (22)$$

(i=1, 2, ..., M₀ on A₀)

なる流れの出入りを生ずるので、これを補正するために 𝕵⁽¹⁾ を船体表面に付加する。これは (13) 式の境界条件 を近似的に満たすことになる。 の(1) は次の式で与えられ る。

$$\sigma_0^{(1)}(i) = -\frac{1}{4\pi} v_n(i) \tag{23}$$

$$(i=1, 2, \dots, M_0 \text{ on } A_0)$$

以上のようにして σ_1, σ_0 の第一近似解 $\sigma_1^{(1)}, \sigma_0^{(1)}$ が求め られた。

(第二段) σ1 の第二近似解 σ1(2) を求める連立方程式は 次のようになる。

$$\sum_{j=1}^{M_{1}} \sigma_{1}^{(2)}(j) A_{1}(ij) - 2\pi g \sigma_{1}^{(2)}(i)$$

$$= B(i) - \sum_{j=1}^{M_{0}} \sigma_{0}^{(1)}(j) A_{0}(ij) \qquad (24)$$

$$(i=1,2,\cdots,M_{1} \text{ on } A_{1})$$

これは(21)式の右辺にのの影響が加えられた形になっ ており、左辺のマトリクスは基本的に第一段のものと同 じである。これを解いて o1⁽²⁾ が求められ, 第一近似と 同様にして σ_0 の第二近似解 $\sigma_0^{(2)}$ を求める。

第三段階以降は第二段階を繰り返すことになる。なお 各段階の終りに計算される圧力抵抗を、収束の判定に用 いる。実際の計算において、第三段階までの計算でほぼ 1パーセント以内の誤差に収束することが確認された。

2.4 二重模型流場 🖓 の計算

本論文における境界層および伴流の計算は、文献13)の 方法に従った積分的解法 である。 二次流を微小と仮定 し、高次の項およびηに関する微係数を省略した運動量 の積分条件式は, Fig.1 に示す直交曲線座標を用いると 以下のようになる。

$$\frac{\partial}{h_1\partial\xi} (U^2\theta_{11}) + \delta_1 U \frac{\partial U}{h_1\partial\xi} - K_1 U^2\theta_{11} = \frac{\tau_{\xi}}{\rho}$$
(25)
$$\frac{\partial}{h_1\partial\xi} (U^2\theta_{21}) - 2K_1 U^2\theta_{21} + K_2 U^2(\theta_{11} + \delta_1)$$

$$= \frac{\tau_{\xi}}{\rho} \tan\beta$$
(26)

このとき ξ は船体表面のポテンシャル流線, η は等ポテ ンシャル線、Cは船体表面の外向き法線を表わし、(u, v,w)をそれぞれの方向成分とする流速ベクトルとする。 また U は境界層外端流速, K_1 は流線の拡縮率, K_2 は 流線の測地的曲率, h_1, h_2 はそれぞれ ξ, η 軸の尺度係 数, τ_{ϵ} は壁面せん断応力の ϵ 方向成分, β は壁面にお ける二次流角である。 さらに運動量厚 $heta_{11}, heta_{21}$, 排除厚 δ_1 は以下のように定義される。

(25)(26) に Ludwieg-Tillmann の局部摩擦則, Head および奥野の Entrainment 式, 二次流運動量モーメン ト式を加え、文献18)の反復的解法を用いて、粘性流場を 解く。 **q** が境界層もしくは伴流中の流れである場合、 以下の流速モデルを用いて u,v を求める。

$$\pm \tilde{\pi} \begin{cases} \frac{u}{U} = \left(\frac{\zeta}{\delta}\right)^m & m = \frac{H-1}{2} \quad H = \frac{\delta_1}{\theta_{11}} (\mathfrak{G} \mathfrak{R} \mathfrak{R}) \\ \frac{u}{U} = 1 - \left(1 - \frac{u_0}{U}\right) \cos^4\left(\frac{\pi\zeta}{2\delta}\right) & (\mathfrak{R} \mathfrak{K}) \end{cases}$$

$$(28)$$

二次流
$$\frac{v}{u} = \tan \beta \left(1 - \frac{\zeta}{\delta} \right)^2 + C \left(\frac{\zeta}{\delta} \right) \left(1 - \frac{\zeta}{\delta} \right)^2$$
 (29)

またヵ方向の微係数を省略した連続の式を積分すること によりwを求める。

$$w(\zeta) = \int_0^\zeta \left(K_1 u - \frac{\partial u}{h_1 \partial \xi} \right) d\zeta + \int_0^\zeta K_2 v d\zeta \quad (30)$$

境界層と伴流中における え。は (28)(29)(30) で求まる (u,v,w) で与えられる。

境界層、伴流の外部はポテンシャル流場と見なすこと ができ、境界層および伴流の排除影響を考慮した速度ポ テンシャルは以下のようになる。

$$\phi = U_{\infty}x - \iint_{A_H} (m_H + m_v) \frac{1}{r} dS_H - \iint_{A_W} m_W \frac{1}{r} dS_W$$
(31)

ここで U_{∞} は一様流速, m_H は Hess & Smith 法で求ま る船体表面上吹き出し分布密度, m。は境界層の排除影 響を表わす吹き出し分布密度, mw は伴流の排除影響を 表わす吹き出し分布密度, A_H は船体表面, dS_H はその 面素, Aw は船尾端から後方の中心線面, dSw はその面 素である。rは各面素の位置から流場内の点までの距離 を表わす。したがってポテンシャル流域におけるす。は、 $\vec{q}_0 = \vec{V}\phi$

(32)

で求められる。

2.5 Head loss の計算法

(5) 式で与える $g\delta H$ を求めるためには、流場内の流 速,および圧力を計算しておく必要がある。境界層外流 域は、ポテンシャル流と見なせるので、そこにおける Head loss は零となる。境界層内部の流速分布,および 圧力分布は、境界層計算で求めた各境界層パラメータを 用いて計算する。船体表面の法線方向の圧力勾配を考え れば(5)式は次のようになる。

$$g\delta H = \frac{U_{\infty}^2}{2} - \left\{ \frac{q_0^2}{2} + \frac{P_e}{\rho} - \int_{\zeta}^{\delta} \frac{\partial}{\partial \zeta} \left(\frac{P}{\rho} \right) d\zeta \right\}$$
(33)

ここで P_e は境界層外端における圧力, δ は境界層厚で あり、 ζ は物体法線方向にとられている。積分式中の圧 力勾配は通常薄い境界層方程式では無視されるが、流体 の遠心力影響を考慮すれば永松¹⁶⁾にならって以下のよう に与えられる。

$$\frac{\partial}{\partial \zeta} \left(\frac{P}{\rho} \right) = \frac{\kappa_{1w}}{1 + \kappa_{1w} \zeta} u^2 + \frac{\kappa_{2w}}{1 + \kappa_{2w} \zeta} v^2 \quad (34)$$

ここで *κ*_{1w}, *κ*_{2w} はそれぞれ *ξ*, η 方向の物体表面の曲率, *u*, *v* は各方向の流速である。

2.6 波高および抵抗成分の計算

船体表面および自由表面上に分布された Rankinesource が求められたところで、波高、および船体に働 く抵抗成分を計算する。波高は(4)式より求める。た だし Φ_w に関する高次の項は省略する。

$$\zeta(x,y) = \frac{1}{2g} (U_{\infty}^{2} - u_{0}^{2} - v_{0}^{2} - 2u_{0}\Phi_{wx} - 2v_{0}\Phi_{wy} - 2g\delta H)$$
(35)

また前節と同様,物体法線方向の圧力勾配を考慮し,船 体表面の圧力 *Pw* は次の式で与えられる。

$$P_{W} = P_{e} - \int_{0}^{\delta} \frac{\partial}{\partial \zeta} \left(\frac{P}{\rho}\right) d\zeta \qquad (36)$$

船体表面の面素内で圧力が一定であるとすれば、圧力抵 抗は次式で計算される。

$$R_{P} = -\sum_{j=1}^{M_{0}} \{P_{W}(i) - P_{0}\} n_{x}(i) \Delta S_{0}(i) \qquad (37)$$

ここで ΔS_0 は面素の面積, n_x は法線の方向余弦, P_0 は 無限遠方の圧力 である。一方, 船体表面の摩擦応力は Ludwieg-Tillmann の局部摩擦則 を 仮定しているので 次式で与えられる。

$$\frac{\tau_{\varepsilon}}{\rho U^2} = 0.123 \times 10^{-0.678H} \left(\frac{U\theta_{11}}{\nu}\right)^{-0.268}, \ \tau_{\eta} = \tau_{\varepsilon} \tan\beta$$
(38)

したがって船体全表面に働く摩擦抵抗 Rf は

$$R_f = \iint_{A_0} \tau_w l_x dS_0, \quad \tau_w = \sqrt{\tau_{\xi}^2 + \tau_{\eta}^2} \quad (39)$$

ここで l_x は合力 τ_w のx方向の方向余弦である。

計算結果と考察

計算の対象船型として,6m 長の Wigley 模型を選ん だ。この模型はよく知られるように,次式で表現される 数学模型である。

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

ただし、*L*=6.0m=長さ、*B*=0.6m=幅、*d*=0.375 m=吃水である。

計算の手順は, Fig.2 に示す流れ図に従って行われる。まず, 船体表面の要素分割を行う。Wigley 模型は 薄い船型の部類に入るので, 水線と横断面における肋骨 線とを用いたメッシュでほど良くパネルに分割できる。



Fig. 2 Flow chart of numerical computation



Fig. 3 Panel division on hull surface and center plane in wake



Fig. 4 Body plan and potential streamlines of Wigley model

Fig. 3 にそのパネル分割の様子を示す。パネルの数は, 船体表面片側で260, 伴流中央面で90 である。次いで, 二重模型近似によるポテンシャル流線の計算を行う。今 回境界層計算のために Fig. 4 に示す 10 本の流線を計算 で求めた。

3.1 境界層と伴流

ポテンシャル計算で求めた 10 本の流線に沿って,前述した計算法によって3次元境界層および伴流の計算を 行った。代表的な流線4本についてその結果を示す。



Fig. 5 Solutions of boundary layer and wake along streamlines for displacement thickness, shape factor and cross flow angle



Fig. 6 Solutions of boundary layer and wake along streamlines for momentum thickness and boundary layer thickness



Fig. 7 Boundary layer edge surface around Wigley model $(R_n=1.201\times10^7)$



Fig. 9 Panel division on free surface

Fig.5 に排除厚 δ_1 , 形状係数 H, 二次流角 β を示す。 Fig.6 に境界層厚 δ ,および運動量厚 θ_{11} を示す。いず れにせよ、境界層計算の反復は4回で約4%以内の誤差 に境界層厚δが収束したので、そこで打切った。Fig.5, Fig.6 の計算結果をみると、対象船型が比較的痩せて単 調な形状であるので,船尾付近における解の変動もさほ ど激しくなく、船尾端における発散も防がれ、伴流まで 連続した結果が得られている。形状係数Hが船尾直後で 急に減少しているのは、境界層から伴流に移行したため である。二次流角 β が船尾直後で符号を負に転じている のは、ビルジ渦による下向流の存在を明らかに示してい る。船体後方における排除厚および運動量厚は、水面に 近いほど発達が顕著である。Fig.7 に境界層外端面の外 観を示すが、全体的に船体後方そして水面に向って広が っている様子が見られる。Fig.8 に静水面における流速 ベクトル 夏。の分布を示す。船尾付近にて境界層が発達 しやがて伴流に移行して行く著しい流速分布の変化の過 程がよく見られる。

3.2 自由表面流

Fig.9 に静水面のパネル分割の様子を示す。静水面の パネル分割の縦方向は q_0 により計算された流線,横方 向は船体の幅方向に平行な直線を用いた。 q_0 による流線 は,船体が前後対称にもかかわらずポテンシャル流の場 合と違い船体前後で対称な流線にはならず,後半部で境 界層と伴流の排除効果により外側に押し出されている。 しかるに,流線の間隔はさほど大きな差もなく比較的均 等に近いので,この流線をそのままパネル分割に用いて も支障はないと判断した。よくやるように船体前後端付 近のパネル分割を細かくすることは,波形計算の振動を 避けるために今回行わなかった。Fig.10 には要素上の 流速 q_0 を各要素列に沿って示してある。 q_0 の変動量は,





Fig. 10 Panel array and computed velocity distributions on free surface $(R_n = 1.201 \times 10^7)$



Fig. 11 Head loss along free surface panel lines $(R_n=1.201\times10^7)$







Fig. 13 Computed and measured wave profiles



Fig. 14 Computed wave pattern

LINE.1(船体表面に最も近い)のものが最も著しい。 Fig.11 に LINE.1 から LINE.3 の Head loss を示す が、LINE.1 の Head loss はかなり大きいのに、LINE. 2 から外では急減している。これは、Head loss の影響 が船体表面の極く近傍だけで大きいことを示している、 と解釈できる。また、LINE.1 の Head loss が船尾付 近でピークをとるのに対し、それ以外のパネル列は船体 後方に向うにつれて徐々に増加している。この傾向は、 Fig.10 の q_0 の変化にちょうど対応している。

3.3 圧力,船側波形,抵抗成分

船体表面の圧力分布を Fig. 12 に示す。今回の方法に よる計算とポテンシャル計算と実験¹⁷⁾との3者の比較を 示している。今回の計算法では船尾に近づくにつれ増加 する圧力損失によりポテンシャル計算との差が開き,実 験値に近づいているといえるが,一致度は今一つ物足り ない感じがする。

一方,船側波形の結果を Fig.13 に示す。今回の計算 とポテンシャル計算とは大部分の範囲でほとんど差がな いが,船尾付近で今回の計算の立上りが低く押えられて いる。これが境界層影響による効果であろう。実験値¹⁷⁾ は今回の計算結果に近い。

Fig. 14 に船体まわりの波紋の計算結果を示す。船尾 付近の盛上りがなだらかなのは、やはり境界層と伴流の 影響によるものであろう。

以上,境界層と伴流の影響を,圧力分布と船側波形で 見てきたわけだが,船側波形では大体良いようだが,圧 力分布では少し物足りない計算になっているようであ る。船体表面の圧力を境界層外端の圧力 P_e から内側へ 勾配で修正する (36) 式を用いたことが原因になってい るかもしれない。

-7-





表面圧力を船体表面に亘って(37)式で積分すれば圧 力抵抗が求まるし、船体表面の摩擦応力を(39)式のよ うに積分すれば摩擦抵抗が求まり、両者の合計から全抵 抗が計算できる。 Fig. 15 に、 それらの抵抗成分の計算 結果を示す。圧力抵抗 Cp のポテンシャル理論値は,波 形造波抵抗曲線17)と同傾向を示すが全体に低目である。 一方今回の方法による圧力抵抗 Cp の曲線は、低速では 曳航試験の造波抵抗17)に近いかそれを上回り、高速では 下回るという交叉した傾向を示した。したがってオー ダー的には実験による造波抵抗に一致するが、フルード 数 0.30 付近の山が小さいことなどの点でもう一つ一致 度はよくない。これら二つの Cp の差が、いわゆる粘性 圧力抵抗に相当する抵抗成分を表わしているわけだが、 前述の圧力分布についての考察を考えると、少し大きく 出すぎているかもしれない。摩擦抵抗についてみると、 Schoenherr の平板公式より少し下回るもののほぼ平 行な曲線が得られた。これは相当平板近似の妥当性につ ながるとみなせれば有益な結果である。全抵抗 Ct はフ ルード数 0.27 以上での盛上りに欠けるが、それより低 速ではよく実験値¹⁸⁾に合っている。このように、境界要 素法と境界層理論とを応用した理論計算で、船体に働く 抵抗がかなりの精度で計算できることが、今回の研究で 分かったことは収穫である。

4 結 言

本研究で得られたことを以下に列記する。

(1) ランキン・ソース法によるポテンシャル造波理 論と積分型境界層理論とを組み合わせて,船体まわりの 自由表面流を解くための数値的解法を工夫した。

(2) 船体まわりの境界層と伴流が自由表面流に及ぼ す影響を,自由表面の境界条件に水頭損失を考慮するこ、 とで,解決できる方法を導いた。

(3) Wigley 模型についての計算結果では、実測の 船側波形と良く合った。とくに、従来のポテンシャル理 論では合いにくい船尾付近でも実測に近づく傾向を得 た。

(4) Wigley 模型についての船体表面圧力分布の計 算結果も,境界層と伴流の影響により実験値に近づいた。

(5) Wigley 模型について, 圧力抵抗, 摩擦抵抗, 全抵抗を計算により求め, それぞれ, 造波抵抗, 平板摩 擦抵抗, 曳引全抵抗とまあまあの一致をみた。

終りに当り,有益な助言と討論を賜った丸尾 孟教授 と鈴木和夫助教授に深甚な謝意を表します。なお,本研 究の一部には文部省科学研究補助金の助成を得たこと と,東大大型計算機センターと横浜国大電子計算機セン ターの計算機を利用したことを付記し,関係各位に感謝 いたします。

参考文献

- 1) Havelock, T. H.: "Ship waves : the Relative Efficiency of Bow and Stern", Proceeding of Royal Society, (A) 149, (1935).
- Inui, T.: "Study on Wave-making Resistance of Ships", 60 th Anniversary Series, Society of Naval Architects of Japan, Vol. 1 (1957).
- Milgram, J. H.: "The Effect of a Wake on the Wave Resistance of a Ship", Journal of Ship Research., Vol. 13, No. 1 (1969).
- Okabe, J. and Jinnaka, T.: "On the Waves of Ships", Report of Research Inst. for Fluid Eng., Kyusyu Univ., 71 (1950).
- Hinatsu, M. and Adachi, H.: "On Evaluation of Wave Resistance Including the Effect of Boundary Layer", Journal of Kansai Society of Naval Architects, Japan, No. 185 (1982).
- Hinatsu, M. and Takeshi, H.: "A Calculation Method for Resistance Prediction Including Viscid-Inviscid Interaction", 2 nd International Symposium on Ship Viscous Resistance, 1985 March, Göteborg, Sweden.
- Larsson, L. and Chang, M. S.: "Numerical Viscous and Wave Resistance Calculations Including Interaction", 13 th Symposium on Naval Hydrodynamics, Tokyo (1980).
- 28) 姫野洋司,高木又男:粘性流体における造波抵抗,日本造船学会論文集,第130号(1971).
- 丸尾 盂,松村純一:粘性流体における船の波お よび造波抵抗,日本造船学会論文集,第 134 号 (1973).
- Mori, K.: "Prediction of Viscous Effects on Wave Resistance of Ship in Framework of Low Speed Wave Resistance Theory", Mem. of the Faculty of Eng. of Hiroshima Univ., Vol. 7, No. 1 (1979).
- Mori, K., Nishimoto, H.: "Prediction of Flow Fields around Ships by Modified Rankine Source Method (1st Report)", Journal of

-8-

Society of Naval Architects of Japan, Vol. 150 (1981).

- 13) 池畑光尚,長瀬 裕,丸尾 孟:境界層理論を改 良した船尾粘性流場の解法,日本造船学会論文 集,第152号 (1983).
- 14) Dawson, C. W.: "A Practical Computer Method for Solving Ship Wave Problems", 2 nd Int. Conf. on Numerical Ship Hydrodynamics, Berkeley (1977).
- 15) 荻原誠功,丸尾 孟:船体まわりの自由表面流れ の非線形計算法,日本造船学会論文集,第157号 (1985).

- Nagamatsu, T.: "Calculation of Ship Viscous Resistance and Its Application,", Journal of Society of Naval Architects of Japan, Vol. 157 (1985).
- 17) 並松正明, 荻原誠功, 田中 拓, 日夏宗彦, 梶谷 尚:Wigley 相似模型の水槽試験結果の評価 3. 船体表面圧力計測結果の検討, 関西造船協会誌, 第 197 号 (1985).
- 18) Kajitani, H.: "A Further Study of ITTC Cooperative Experiments-Some Analyses (Wigley model) & Exp. Plannings (S 60 model)--", prepared for 1st Meeting at SSPA R&F C, 21-23 March 1985, (unpublished).

A Boundary-Element Method for Calculating Free-Surface Flows Around a Yawed Ship*

By Yusuke TAHARA (Member)**

The free-surface flows around a yawed ship in steady motion are analyzed by the boundary-element method. The solution is obtained by a distribution of singularities on the ship surface and on the undisturbed free surface. The total velocity potential is divided into two parts, i.e., the double-model lifting-flow potential and the free-surface-flow potential. The double-model lifting-flow potential is expressed by source and doublet distributions, and determined with a Dirichlet-type boundary condition. The free-surface-flow potential is expressed by a simple Rankine-type singularity and determined with linearized boundary conditions applied on the free surface. An overview is given for the present approach, and numerical results are presented for the Wigley hull, including comparisons with available experimental data.

Keywords : Boundary-element Method, Yawed Ship, Ship Wave

1. Introduction

An important factor in the design of ships is accurate determination of the hydrodynamic forces acting on the hull. This is especially true for a ship motion with yaw angle. Many of recent works on this problem have been focused on the analysis of the sailing yacht, whose maneuverability totally relies on its hydrodynamic design. The present study is central to the aforementioned problems, i.e., it is concerned with the development of a numerical approach for calculating free-surface flows around a yawed ship in steady motion.

Considerable effort has been put forth in the investigation of ship-wave related problems. Recent work on this problems has focused on the solution of the so-called Neumann-Kelvin problem using both Rankine-and Havelock-source approaches. A boundary-element method proposed by Dawson¹⁾ has been used by many researchers, and found to be a very useful design tool. Since Dawson's pioneer work, many extensions of this method had been done to calculate more complicated free-surface flows, including nonlinear effect (e.g., see Ogiwara and Maruo²), Kim and Lucas³) or the viscous effects (e.g., see Ikehata and Tahara⁴).

The first extension of the Dawson-type method to lifting flow was presented by Xia and Larsson⁵⁾. In this work, the double-model lifting flow was solved by the method of Hess⁶⁾, which was a extension of Hess and Smith⁷⁾ to include lifting effect. The numerical algorithm of this work was basically same as that of Dawson's original method, and the numerical results presented for a sailing yacht were shown to be very promising.

Rosen⁸⁾ developed another Dawson-type method, i.e., the SPLASH computer code. This method is an extended version of the basic panel method of Maskew^{9),10)} originally developed for the prediction of subsonic aerodynamic flows about arbitrary configurations, in order to include the free-surface effect. In the report¹¹⁾, the numerical results were presented for a 12m sailing yacht, *the Stars* &

^{*} Read at the Spring Meeting of Kansai Society of Naval Architects, Japan, May, 22, 1992, Received March 25, 1992

^{**} Department of Mechanical Engineering, Institute of Hydraulic Research, The University of Iowa

Stripes, including comparisons with the experimental data in the towing tank. This method was also used to calculate nonlifting flow in the work of Tahara et al.¹²⁾, which is concerned with an interactive approach for calculating ship boundary layers and wakes with free-surface effect.

On the other hand, Maruo and Song¹³⁾ applied a new slender-ship formulation to the steady ship motion with yaw angle. In this work, the integral equation is simplified using an asymptotic expression of the Kelvin-source around ship track. Numerical solutions were obtained for several type of ships without lifting effect. It was noted that solutions for yawed ship motion can be obtained without lifting effect, however some difficulties are observed bcause of it.

In the above-mentioned studies, it appeared that lifting effect on the free-surface flow is not small, and is very important for accurate estimation of the hydrodynamic forces acting on the hull. Here we note that the lifting-flow solution method of Rosen⁸⁾ is more suitable for the three-dimensional case than that of Xia and Larsson⁵⁾, for the treatment of boundary and Kutta conditions. Furthermore, the method of Rosen⁸⁾ uses a doublet distribution to express the free-surface-flow potential, however a simple Rakine-type source distribution would be more suitable for the computation, because of its weaker singularity.

Hence, in this paper a method proposed is an extension of Dawson's method¹⁾, in order to include lifting effect. The total velocity potential is first divided into two parts, i.e., the double-model lifting-flow potential and the free-surface-flow potential. The double-model lifting-flow potential is expressed by the sources and doublets distributed on the body and wake surfaces, and determined by the method of Maskew⁹⁾. The free-surface-flow potential is expressed by a simple Rankine-type source distribution on the undisturbed free surface, and source and doublet distributions on the body and wake surfaces. The linearized free-surface boundary condition is used to determine the freesurface-flow potential. In the following, an overview is given for the present numerical approach, and results are presented for the Wigley hull, including comparisons with available experimental data, which validate the overall approach and enable the evaluation of lifting effect on the hydrodynamic forces acting on the hull.

2. Governing Equations

Let us consider a ship fixed in the uniform onset flow $U_{\infty} = (U, V, 0)$ as depicted in Fig 1. Take the Cartesian coordinate system with the origin on



Fig. 1 Coordinate system

the undisturbed free surface, x and y axes on the horizontal plane, and z axis directed vertically upward. Since the fluid is assumed to be inviscid and incompressible and its motion irrotational, the velocitty field can be defined as

$$u(x,y,z) = \Phi_x, \quad v(x,y,z) = \Phi_y, \quad w(x,y,z) = \Phi_z$$
.....(1)

where u, v, and w are the velocity components in the x, y, and z directions respectively, and $\Phi(x, y, z)$ is the velocity potential which satisfies the Laplace equation

in the fluid domain. On the free surface $z = \eta(x, y)$, the kinematic and dynamic boundary conditions are

$$\boldsymbol{\Phi}_{\boldsymbol{x}}\boldsymbol{\eta}_{\boldsymbol{x}} + \boldsymbol{\Phi}_{\boldsymbol{y}}\boldsymbol{\eta}_{\boldsymbol{y}} = \boldsymbol{\Phi}_{\boldsymbol{z}} \quad (\boldsymbol{z} = \boldsymbol{\eta}) , \qquad \cdots \cdots \cdots (3a)$$

$$\frac{1}{2g} [\boldsymbol{\Phi}_{\boldsymbol{x}}^{2} + \boldsymbol{\Phi}_{\boldsymbol{y}}^{2} + \boldsymbol{\Phi}_{\boldsymbol{z}}^{2} - |\boldsymbol{U}_{\boldsymbol{\omega}}|^{2}] + \boldsymbol{\eta} = 0 \quad (\boldsymbol{z} = \boldsymbol{\eta}) \cdots \cdots (3b)$$

where g denotes the gravitational constant. Assuming $\eta(x, y)$ and its derivatives to be small and applying the Taylor-series expansions about the undisturbed free surface, we have

$$\begin{split} & \varphi_{x}\eta_{x} + \varphi_{y}\eta_{y} - \varphi_{z} - \varphi_{zz}\eta = 0 \quad (z=0) \quad \dots \dots \dots (4a) \\ & \eta + \frac{1}{2g} [\varphi_{x}^{2} + \varphi_{y}^{2} + \varphi_{z}^{2} - |U_{\infty}|^{2}] \\ & + \frac{1}{g} [\varphi_{x}\varphi_{xz} + \varphi_{y}\varphi_{yz} + \varphi_{z}\varphi_{zz}]\eta = 0 \quad (z=0) \\ & \dots \dots \dots (4b) \end{split}$$

where terms in η^2 and higher are neglected. Here divide the total potential Φ into two parts as follows:

$$\Phi(x, y, z) = \Phi_0(x, y, z) + \Phi_1(x, y, z) \cdots \cdots \cdots \cdots (5)$$

where Φ_0 expresses the double-model lifting flow and Φ_1 the free-surface flow. Note that Φ_0 is known a-priori if the ship geometry is given. Substituting (5) into (4) we have

where

 $D_1(x, y)$ and $D_2(x, y)$ in (7) are nonlinear terms in Φ_1 . Letting $D_1(x, y)$ and $D_2(x, y)$ be zero to linearize the boundary conditions, and replacing η in (6a) by (6b), we obtain

Tracing a flow particle along the double-modelflow streamline (l), we have the following relation:

Applying (9) to (8), we get

$$\phi_{0l}^{2}\phi_{1ll}^{2} + 2\phi_{0l}\phi_{0ll}\phi_{1l} + g\phi_{1z}^{2} = -\phi_{0l}^{2}\phi_{0ll} \quad (z=0)$$
.....(10)

for the boundary condition which Φ_1 satisfies on the undisturbed free-surface S_f . This is the same as that derived by Dawson¹⁾.

On the other hand, the velocity potential Φ_0 for

the double-model lifting flow is determined in the following manner. Consider a double body immersed in a uniform onset flow $U_{\infty} = (U, V, 0)$ with the velocity potential Φ_{0} . We assume the existence of the velocity potential Φ_{0} in the field and Φ_{0i} inside the body. Also assume that the wake surface S_{w} has vanishing thickness. Applying Green's Theorem to the inner and outer regions and combining the resulting expressions, the velocity potential at a point P on the inside body surface may be written as

$$4\pi \Phi_{0P} - \int_{S_{b}-P} (\Phi_{0} - \Phi_{0i}) \left(\boldsymbol{n} \cdot \nabla \left(\frac{1}{r} \right) + \overline{\boldsymbol{n}} \cdot \nabla \left(\frac{1}{\overline{r}} \right) \right) dS$$

+ $2\pi (\Phi_{0} - \Phi_{0i})_{P}$
- $\int_{S_{w}} (\Phi_{0u} - \Phi_{0u}) \left(\boldsymbol{n} \cdot \nabla \left(\frac{1}{r} \right) + \overline{\boldsymbol{n}} \cdot \nabla \left(\frac{1}{\overline{r}} \right) \right) dS$
- $\int_{S_{b}} \left(\frac{1}{r} + \frac{1}{\overline{r}} \right) \boldsymbol{n} \cdot (\nabla \Phi_{0i} - \nabla \Phi_{0}) dS + 4\pi \phi_{\infty P}$
......(11)

where $S_b - P$ signifies that the point P is excluded from the surface integral over the body surface S_b , and U and L denote the upper and lower surfaces of S_w , r the distance between P and a singularity point, \dot{r} the distance between P and an image of the singularity point about the symmetric plane of the body, $\mathbf{n} = (n_x, n_y, n_z)$ the unit outward normal vector, and $\bar{\mathbf{n}}$ the unit outward normal vector at a image point of the singularity. Equation (11) gives the total potential at the interior point P as the sum of perturbation potentials due to a noral doublet distribution of strength $\boldsymbol{\Phi}_0 - \boldsymbol{\Phi}_{0i}$ on S_b and $\boldsymbol{\Phi}_{0U} - \boldsymbol{\Phi}_{0L}$ on S_w respectively, and a source distribution of strength $\mathbf{n} \cdot (\nabla \boldsymbol{\Phi}_{0i} - \nabla \boldsymbol{\Phi}_0)$ on S_b . Here setting $\boldsymbol{\Phi}_{0i} = \boldsymbol{\phi}_{\infty P} = \boldsymbol{\Phi}_{0P}$, we have

where ϕ is the perturbation potential on the exterior surface, i.e., $\phi = \phi_0 - \phi_{\infty}$. Then equation (12) can be rewritten as

$$0 = -\int_{\mathcal{S}_{b}-P} \mu_{0b} \left(\boldsymbol{n} \cdot \nabla \left(\frac{1}{r} \right) + \overline{\boldsymbol{n}} \cdot \nabla \left(\frac{1}{r} \right) \right) dS + 2\pi \mu_{0bP}$$

where μ_{0b} and σ_{0b} are doublet and source densities respectively. The wake doublet density μ_{0w} is determined by the Kutta condition. Since steady flow is considered here, μ_{0w} can be constant along the wake streamline, and given by

where NU and NL refer to the upper and lower surfaces of the body at the trailing edge. This implies the zero-load condition. In equation (13), let the source density σ_{0b} be

Thus this is known a-priori if the double-model geometry is given. Now we have an integral equation (13) to determine μ_{0b} . The velocity potential for the double-model lifting flow is given by

with

$$\phi_{\infty} = U \cdot x + V \cdot y \qquad \qquad \cdots \cdots \cdots (16b)$$

at a field point P(x, y, z).

In this study, the free-surface potential Φ_1 is expressed by the Rankine-type sources distributed on S_f and sources and doublets distributed on S_b snd S_w . At a field point P(x, y, z), the velocity potential for the free-surface flow is

where μ_{1b} and μ_{1w} are the doublet densities, and σ_{1b} and σ_{1f} the source densities. The wake doublet density μ_{1w} is given in similar manner as that for μ_{0w} , i.e.,

in order to satisfy the Kutta condition. On the body surface S_b , let the source density σ_{1b} be

where v_n is normal velocity induced by σ_{1f} distributed on S_f . Then the boundary condition which Φ_1 satisfies about the body surface S_b is given similarly to the double-model flow as

at a point P on the inside body surface.

After Φ_0 and Φ_1 are determined, the pressure P in the flow field is

$$p - p_{\infty} = \frac{1}{2} \rho [|U_{\infty}|^2 - \phi_{0x}^2 - \phi_{0y}^2 - \phi_{0z}^2 - 2\phi_{0x}\phi_{1x} - 2\phi_{0y}\phi_{1y} - 2\phi_{0z}\phi_{1z}] \dots (21)$$

where p_{∞} is the pressure in the uniform onset flow. In equation (21), nonlinear terms in Φ_1 are neglected. The hydrodynamic forces acting on the body surface are then given by

Similarly, the expression of the free-surface elevation is

3. Numerical Approach

In the following, the overview is given for the present numerical approach. First, the boundary surfaces S_f , S_b and S_w are divided into quadrila-

teral panels, and the source and doublet densities are approximated by a constant value on each panel. Equation (15) gives the source density $\sigma_{0b}(j)$ on the *j*-th panel as

$$\sigma_{0b}(j) = -\frac{1}{4\pi} \boldsymbol{n}(j) \cdot \boldsymbol{U}_{\infty} \quad (j=1, N_0) \cdot \cdots \cdot (24)$$

where N_0 is the number of panels on the body surface S_b . Replacing the integration of (13) by summation, we have the discretized boundary condition at the centroid of the *i*-th panel as follows:

or

$$0 = \sum_{j=1}^{N_0} A_4(ij) \mu_{0b}(j) + 2\pi \mu_{0b}(i) + \sum_{j=1}^{N_0} A_3(ij) \sigma_{0b}(j)$$

(*i*=1, N₀)(25b)

where N_{0w} is the number of panels on the wake surface S_w . Solutions to the matrix equation (25b) and equation (14) give doublet densities $\mu_{0b}(j)$ and $\mu_{0w}(j)$. At a field point P(x, y, z), the velocity potential for the double-model lifting flow is thus given by

In the present study, the wake surface is assumed to be parallel to the uniform onset flow, and placed at the stern edge and keel line, in which vortices are assumed to be generated.

On the other hand, equations (10) and (18) through (20) give the matrix equations to determine Φ_1 , i.e.,

or

$$0 = \sum_{j=1}^{N_0} C_{10}(ij) \mu_{1b}(j) + 2\pi \mu_{1b}(i) + \sum_{j=1}^{N_0} C_8(ij) \sigma_{1b}(j) + \sum_{j=1}^{N_0} C_9(ij) \sigma_{1f}(j) \quad (i = 1, N_0) \quad \dots \quad (27d)$$

where N_1 is the number of panels on the free surface S_f . Combining (27) we have a full matrix equation whose solutions give the source densities $\sigma_{1b}(j)$, $\sigma_{1f}(j)$ and the doublet densities $\mu_{1b}(j)$, $\mu_{1w}(j)$. Then the velocity potential for the free-surface flow is given by

$$\Phi_{1}(x, y, z) = \sum_{j=1}^{N_{0}} E_{1}(j)\sigma_{1b}(j) + \sum_{j=1}^{N_{1}} E_{2}(j)\sigma_{1f}(j) \\
+ \sum_{j=1}^{N_{0}} E_{3}(j)\mu_{1b}(j) + \sum_{j=1}^{N_{0}w} E_{4}(j)\mu_{1w}(j) \dots (28)$$

at a field point P(x, y, z).

After Φ_0 and Φ_1 are determined, $\eta(x, y)$ is calculated by equation (23). In addition, assuming the pressure is constant on each panel, we have the following expression from (22):

for calculation of the hydrodynamic forces acting on the ship surface.

In the present study, the radiation condition for the free-surface flow is satisfied numerically, using the similar manner as that of Dawson¹⁾. However a more generalized expression is used in order to utilize arbitrary panel arrangement on the free surface. In the work of Dawson, $\boldsymbol{\Phi}_{111}$ is evaluated by upstream differentiation to express the radiation condition. Here introducing a nonorthogonal coordinate system with $\boldsymbol{\xi}$ in the longitudinal and $\boldsymbol{\zeta}$ in the transverse directions, $\boldsymbol{\Phi}_{111}$ is written as

In this study, $(\varPhi_{1l})_{\xi}$ and $(\varPhi_{1l})_{\zeta}$ are evaluated by the four-point upstream differentiation in the whole free-surface domain. A similar approach was used in the work of Xia et al.⁵⁾ or Rosen⁸⁾.

4. Results and Discussion

In the following, first the computational panel arrangement (Fig. 2) and conditions are described. Then some example results are presented and



Fig. 2 Panel arrangement on the free surface

discussed for zero yaw angle, followed by those for nonzero yaw angle, including whenever possible comparisons with available experimental data. Computed wave elevations alongside the model are shown in Figures 3 and 4. Fig. 5 provides detailed comparisons of the free-surface profiles for zero yaw angle, between computed and measured results. Computed pressure distributions on the hull for zero yaw angle are compared with measurements in Figures 6 and 7. Fig. 8 provides the comparisons of wave-making resistance for zero yaw angle, between computed and measured results. Computed wave elevations alongside the body for nonzero yaw angle are compared with measurements in Fig. 9, and detail comparisons of the free-surface profiles are shown in Fig. 10. Global and local comparisons of the wave contours, and perspective views of the free-surface are shown in Figures 11 through 13. Lastly, computed hydrodynamic forces acting on the hull for nonzero yaw angle are shown in Fig. 14. In the presentation of the results, variables are non-dimensionalized using the ship length L, the free-stream speed $|U_{\infty}|$ and fluid density ρ .

The surface geometry of the Wigley hull is given by the equation

where b=B/2 is half breadth and d is the draft at the still waterline. A model of dimensions L=2.0m, B=0.2m, d=0.125m has been employed for experiments in the towing tank of Yokohama National University, (hereafter referred to as YNU), at Froude number $F_r = |U_{\infty}| / \sqrt{Lg} = 0.267$ and yaw angle $\alpha = 0^\circ$ and 10° .

The free-surface panelization is shown in Fig.2. 2600 panels are distributed over the free surface, and 600 over the ship hull for a total number of 3200 panels. The panelization covers an area corresponding to one-half ship length upstream of the bow, one ship length in the transverse direction, and one ship length downstream of the stern. This panel arrangement was judged optimum based on panelization dependency tests.

Figures 3 and 4 show the free-surface profiles



Fig. 3 Wave profile alongside the model $(Fr=0.316 \ \alpha = 0^{\circ})$



Fig. 4 Wave profile alongside the model $(Fr=0.267 \ \alpha=0^{\circ})$



Fig. 5 Wave profile in transverse section (Fr=0.267 $\alpha = 0^{\circ}$)

alongside the model for $\alpha = 0^{\circ}$, $F_r = 0.316$ and 0.267 respectively. On the figures, IHI refers to the measurements with the 6-m model at the Ishika-wajima-Harima Heavy Industries Co., Ltd., and SRI the measurements with the 4-m model at the Ship Research Institute of Japan. Generally good agreement is observed between computed and

measured results. The computation gives slightly lower wave elevations near the bow, which is likely due to the lack of nonlinear effects. Detailed comparisons of the free-surface profiles between computation and experiment are shown in Fig.5, for $\alpha = 0^{\circ}$, $F_r = 0.267$. Results are plotted in the y-direction at twelve transverse sections, in which



Fig. 6 Girthwise distribution of pressure (Fr=0.316 $\alpha = 0^{\circ}$)



Fig. 7 Girthwise distribution of pressure (Fr=0.250 $\alpha = 0^{\circ}$)

x=0.0 and 1.0 correspond to the FP. and AP., respectively. Although details in the measurements are not exactly reproduced by the computation, overall agreement between the two results is good.

Figures 6 and 7 show the girthwise distribution of pressure coefficient at twelve transverse sections for $\alpha = 0^{\circ}$, $F_r = 0.316$ and 0.250 respectively. The pressure coefficient is defined as

The girth length is measured from the waterline to the keel, and non-dimensionalized using total girth length at each station. The experimental data with



Fig. 8 Wave resistance coefficient ($\alpha = 0^{\circ}$)





Fig. 9 Wave profile alongside the model $(Fr=0.267 \ \alpha=10^\circ)$

the 6-m model of SRI are included in the figures. It is noted from these figures that the computed pressure distribution is in good agreement with the experimental data. Surface integration of the pressure over the hull gives the wave-making resistance. Fig.8 provides the comparisons between computed results and measurements¹⁴⁾. The computation gives slightly higher values than the





Fig.10 Wave profile in transverse section (Fr=0.267 α =10°) (a) Starboard side (b) Port side

experiment around $F_r = 0.250$, however the positions of humps and hollows appear to be in very good agreement.



(a) Fr=0.316 α=0°



(b) Fr=0.267 α=0°



(c) Fr=0.267 α=10°

Fig.11 Computed wave contours in global view (a) $Fr=0.316 \quad \alpha = 0^{\circ}(b) \quad Fr=0.267 \quad \alpha = 0^{\circ}(c)$ $Fr=0.267 \quad \alpha = 10^{\circ}$

Fig.9 shows the free-surface profiles alongside the model for $\alpha = 10^{\circ}$, $F_r = 0.267$. Note that the port and starboard sides of the hull are the pressure and suction sides respectively (see Fig.1). The experimental data with the 2-m model of YNU are included in the figures. Except for the bow region, computed results are in good agreement with the measurements. For the starboard side, a slight deviation is observed at the bow region in which there might be flow separation in the experiment. The computed bow wave for the port side has a lower predicted amplitude than the experimental data, however the phase of the wave is similar. This may be due, in part, to nonlinear effects. Detailed comparisons of the free-surface profiles between computed and measured results are shown in Fig. 10. Generally, agreement between the



Fig.12 Computed wave pattern (a) Fr=0.316 $\alpha = 0^{\circ}$ (b) Fr=0.267 $\alpha = 0^{\circ}$ (c) Fr=0.267 $\alpha = 10^{\circ}$

two results is good, however some discrepancies can be observed in the starboard side, which is likely due to the flow separation or viscous effects in the experiment.

Figures 11-a through -c provide the global view of the free-surface contours. In these figures, contour interval is 0.002, and solid and dotted lines signify positive and negative contours res-



(a) Experiment



(b) Computation

Fig.13 Wave contours in local view (Fr=0.267 α =10°) (a) Experiment (b) Computation



Fig.14 Computed longitudinal and lateral force coefficients (Fr=0.267)

pectively. In the present calculation, the bow and the stern waves are clearly simulated. For nonzero yaw angle, the wave profiles are completely altered. This is more clearly observed in the perspective view of the free-surface profiles, provided in Fig.12. Figures 13-a and -b show the local comparisons of the free-surface contours between calculated and measured results for $\alpha = 10^{\circ}$, $F_r = 0.267$. A similar configuration in the crests and troughs of the wave patterns are observed, however the computation shows some difficulties in predicting the complicated wavy effects, especially in the stern region. This is mainly due to the lack of viscous effects, and may be partially due to the present panel resolution.

Lastly, Fig.14 shows the computed longitudinal and lateral hydrodynamic forces acting on the hull. As α increases the longitudinal force increases, due to the induced drag. This phenomena is not observed in the result of work of Maruo et al.¹³⁾, in which lifting effect is not considered. The lateral force increases fairly rapidly as α increases. Unfortunately, there are no available experimental data for comparison. Also it would be very difficult to evaluate the present results, since no viscous effect is considered in the theory. However, these computed results appear to be reasonable based on physical intuition.

Conclusions

The present work demonstrates the feasibility of a boundary-element method for calculating the free-surface flows around a yawed ship in steady motion. The results presented for the Wigley hull are very encouraging. In many respects, agreement between the present results and the experimental data is satisfactory. Also it appears that the widely used Dawson-type method fot nonlifting flow can be extended to lifting flow with relatively small modifications. However a complete evaluation of the present method was not possible, due to the limited available experimental data.

Finally, some of the issues that need to be addressed while further developing and validating the present approach are as follows : further assessment of the most appropriate free-surface conditions; the inclusion of nonlinear free-surface effects ; and more complete evaluation of the present method with experimental data. Also it is of great interest to apply this method to a more complicated hull geometry, such as a sailing yacht with a keel winglet.

5. Acknowledgement

The author wishes to thank Prof. M. Ikehata at Yokohama National University for the donation of experimental data. Also his thanks are extended to Dr. Y. Toda at Osaka University, and Prof. L. Landweber, Prof. F. Stern and many graduate students at the Iowa Istitute of Hydraulic Research for their valuable discussions and encouragements.

It is noted that the numerical work in this study had been carried out by the use of CRAY-Y/MP, and again the author expresses his thanks to the Weeg Computing Center of The University of Iowa for student allocation of supercomputing hours at the National Center for Supercomputing Applications.

References

- Dawson, C. W., "A Practical Computer Method for Solving Wave Problems," Proc. 2nd International Conference on Numerical Ship Hydrodynamics, Berkeley, 1977, pp.30-38.
- Ogiwara, S. and Maruo, H., "A Numerical Method of Non-linear Solution for Steady Waves around Ships," J. Society of Naval Architects of Japan, Vol. 157, 1985, pp. 35-47.
- Kim, Y. H. and Lucas, T., "Nonlinear Ship Waves," Proc. 18th Symposium on Naval Hydrodynamics, Ann Arbor, 1990, pp.439-452.
- IKehata, M. and Tahara, Y., "Influence of Boundary Layer and Wake on Free Surface Flow around a Ship Model," J.Society of Naval Architects of Japan, Vol. 161, 1987, pp.49-57.
- 5) Xia, F. and Larsson, L., "A Calculation Method for the Lifting Potential Flow Around Yawed, Surface-Piercing 3-D Bodies," Proc. 16th Symposium on Naval Hydrodynamics, Berkeley, 1986, pp.583-597.
- Hess, J. L., "Calculation of Potential Flow about Arbitraty Three-Dimensional Lifting Bodies," Douglas Report N00019-71-C-0524, 1972, Douglas Aircraft Company, USA.
- 7) Hess, J. L., and Smith, A. M. O., "Calcu-

lation of Non-Lifting Potential Flow about Arbitrary Three-Dimensional Bodies," Douglas Report No. E S 40622, 1962, Douglas Aircraft Company, USA.

- Rosen, B., "SPLASH Free-Surface Code : Theoretical / Numerical Formulation," South Bay Simulations Inc., Babylon, NY, 1989.
- Maskew, B., "Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods," J. Aircraft, Vol.19, No.2, 1982, pp.157-163.
- Maskew, B., "A Computer Program for Calculating the Non-Linear Aerodynamic Characteristics of Arbitrary Configurations, "NASACR-166476, 1982.
- Boppe, C. W., Rosen, B. S., Laiosa, J. P. and Chance, B., Jr., "Stars & Stripes '87: Computational Flow Simulations for Hydrodynamic Design, "The Eighth Chesapeake Sailing Yacht Symposium, Annapolis, MD., 1987.
- 12) Tahara, Y., Stern, F. and Rosen, B., "An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," Proc. 18th Symposium on Naval Hydrodynamics, Ann Arbor, 1990, pp.699-720, also to appear J. Computational Physics.
- Maruo, H. and Song, W. S., "Numerical Appraisal of the New Slender Ship Formulation in Steady Motion," Proc. 18th Symposium on Naval Hydrodynamics, Ann Arbor, 1990, pp.239-258.
- 14) Kajitani, H., "A Further Study of ITTC Cooperative Experiments-Some Analysis (Wigley model) & Exp. Planning (S 60 model)-," presented for 1st Meeting at SSPA R & FC, 21-23 March, 1985 (unpublished).

Discussion

[Discussion] (Memorial University of Newfoundland) N. Bose

Did the author check the actual pressure coefficient at the trailing edge on "upper" and "lower" surfaces to satisfy the Kutta condition. Due to crossflow this sometimes requires an iterative techique, see for example Kerwin et al. (1987) ; Hoshino (1989).

Kerwin, J. E., Kinnas, S. A., Lee, J. -T. and Shih, W. -Z.

A Surface panel method for the hydrodynamic abalysis of ducted propellers, JSNAME, 95 1987 Hoshono, T.

Hydrodynamic Analysis of Propellers in Steady Flow Using a Surface Panel Method, Journal of the Society of Naval Architects of Japan Vol 165, 1989

[Author's Reply]

As shown in the following Fig. A, the Kutta condition at the trailing edge is satisfied. In the present study, a simplified numerical Kutta Condition, i.e., fixed wake sheet, was used, assuming that the crossflow to the wake sheet whould be small. However, an iterative procedure to locate the wake sheet more exactly would be preferable and will be considered in future work. Thank you very much for your valuable discussion.



Fig.A Computed pressure profile on the hull surface (Fr=0.267 $\alpha = 10^{\circ}z=0.5d$)

An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number

Y. TAHARA, F. STERN

Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa 52242

AND

B. ROSEN

South Bay Simulations Inc., 44 Sumpwams Ave., Babylon, New York 11702

Received August 1, 1990; revised December 11, 1990

An interactive approach is set forth for calculating ship boundary layers and wakes for nonzero Froude number. The Reynolds-averaged Navier–Stokes equations are solved using a small domain with edge conditions matched with those from a source-doublet Dawson method solved using the displacement body. An overview is given of both the viscous- and inviscid-flow methods, including their treatments of the free-surface boundary conditions and interaction procedures. Results are presented for the Wigley hull, including comparisons for zero and nonzero Froude number and with available experimental data and the inviscid-flow results, which validate the overall approach and enable an evaluation of the wave-boundary layer and wake interaction. © 1992 Academic Press, Inc.

INTRODUCTION

The interaction between the wavemaking of a ship and its boundary layer and wake is a classic and important problem in ship hydrodynamics. Initially, the interest was primarily with viscous effects on wave resistance and propulsive performance due to the lack of Reynolds number (Re) similarity in model tests. More recently, also of interest are the wave-boundary layer and wake interaction effects on the details of ship wakes and wave patterns due to the advent of satellite remote sensing. The present study is central to the aforementioned problems; i.e., it concerns the development of an interactive approach for calculating ship boundary layers and wakes for nonzero Froude number (Fr). Thus, both the effects of wavemaking on the boundary layer and wake and, vice versa, the effects of the boundary layer and wake on wavemaking are included in the theory, although the focus here is somewhat more on the former.

Historically, inviscid-flow methods have been used to calculate wavemaking and viscous-flow methods the boundary layer and wake, in both cases, without accounting for the

interaction. Recent work on wavemaking has focused on the solution of the so-called Neumann-Kelvin problem using both Rankine- and Havelock-source approaches. Method implementing these approaches were recently competitively evaluated and ranked by comparing their results with towing-tank experimental data [1]. In general, the methods underpredicted the amplitude of the divergent bow waves, were lacking in high wave-number detail in the vicinity of the bow-wave cusp line, and overpredicted the amplitudes of the waves close to the stern. These difficulties were primarily attributed to nonlinear and viscous effects. The methods using the Havelock-source approach generally outperformed those using the Rankine-source approach, except with regard to the near-field results (i.e., within one beam length of the model) for which one of the latter methods [2] was found to be far superior.

Considerable effort has been put forth in the development of viscous-flow methods for ship boundary layers and wakes. Initially, three-dimensional integral and differential boundary-layer equation methods were developed; however, these were found to be inapplicable near the stern and in the wake. More recently, efforts have been directed towards the development of Navier-Stokes (NS) and Reynolds-averaged Navier-Stokes (RANS) equation methods; hereafter both of these will simply be referred to as RANS equation methods. At present, the status of these methods is such that practical ship geometries can be considered, including complexities such as appendages and propellers. Comparisons with experimental data indicate that many features of the flow are adequately simulated; however, turbulence modeling and grid generation appear to be pacesetting issues with regard to future developments (see, e.g., the review by Patel [3] and the Proceedings of the 5th International Conference on Numerical Ship Hydrodynamics $\lceil 4 \rceil$).

0021-9991/92 \$3.00 Copyright © 1992 by Academic Press, Inc. All rights of reproduction in any form reserved. Relatively little work has been done on the interaction between wavemaking and boundary layer and wake. Most studies have focused separately on either the effects of viscosity on wavemaking or the effects of wavemaking (i.e., waves) on the boundary layer and wake. Professor Landweber and his students have both demonstrated experimentally the dependence of wave resistance on viscosity and shown computationally that by including the effects of viscosity in inviscid-flow calculations of wave resistance better agreement with experimental data is obtained (most recently, [5]). Such effects have been confirmed by others, including other more detailed aspects of the flow field such as surface-pressure distributions and wave profiles and patterns [6].

Most studies concerning the effects of waves on boundary layer and wake have been of an approximate nature, utilizing integral methods and assuming small crossflow conditions (see Stern [7] for a more complete review, including references). In [7, 8], experiment and theory are combined to study the fundamental aspects of the problem utilizing a unique, simple model and computational geometry, which enabled the isolation and identification of certain important features of the wave-induced effects. In particular, the variations of the wave-induced piezometricpressure gradients are shown to cause acceleration and deceleration phases of the streamwise velocity component and alternating direction of the crossflow, which results in large oscillations of the displacement thickness and wallshear stress as compared to the no-wave condition. For the relatively simple geometry studied, first-order boundarylayer calculations with a symmetry-condition approximation for the free-surface boundary conditions were shown to be satisfactory; however, extensions of the computational approach for practical geometries were not successful [9].

Miyata *et al.* [10] and Hino [11] have pursued a comprehensive approach to the present problem in which the NS equations (sub-grid scale and Reynolds averaged, respectively) are solved using a large domain with approximate free-surface boundary conditions. In both cases, the basic algorithms closely follow those of MAC [12] and SUMMAC [13]. However, [10] uses a time-dependent free-surface conforming grid, whereas [11] uses a fixed grid which does not conform to the free surface. The results from both approaches are promising, but, thus far, have had difficulties in accurately resolving the boundary-layer and wake regions and, in the case of [10], have been limited to low Re.

The present interactive approach is also comprehensive. Two of the leading inviscid- [2] and viscous-flow [14] methods are modified and extended for interactive calculations for ship boundary layers and wakes for nonzero Fr. The interaction procedures are based on extensions of those developed by one of the authors for zero Fr [15]. The work of [7, 8, 15] is precursory to the present study. Also, it should be mentioned that the present study is part of a large project concerning free-surface effects on boundary layers and wakes. Some of the related studies under this project will be referenced later.

In the following, an overview is given of both the viscousand inviscid-flow methods, with particular emphasis on their treatments of the free-surface boundary conditions and the interaction procedures. Results are presented for the Wigley hull, including comparisons for zero and nonzero Fr and with available experimental data and inviscid-flow results, which validate the overall approach and enable an evaluation of the wave-boundary layer and wake interaction. In the presentation of the computational methods and results and discussions to follow, variables are either defined in the text or in the Appendix and are nondimensionalized using the ship length L, freestream velocity U_o , and fluid density ρ .

COMPUTATIONAL METHODS

Consider the flow past a ship-like body, moving steadily at velocity U_o , and intersecting the free surface of an incompressible viscous fluid. As depicted in Fig. 1, the flow field can be divided into four regions in each of which different or no approximations can be made to the governing RANS equations: region 1 is the inviscid flow; region 2 is the bow flow; region 3 is the thin boundary layer; and region 4 is the thick boundary layer and wake. The resulting equations for regions 1 and 3 and their interaction (or lack of one) are well known. Relatively little is known about region 2. Recent experiments concerning scale effects on near-field wave patterns have indicated a Re dependency for the bow wave both in amplitude and divergence angle [16]; however, this aspect of the problem is deferred for later study. Herein, we are primarily concerned with the flow in region 4 and its interaction with that in region 1. As discussed earlier, the description of the flow in region 4 requires the solution of the complete RANS equations (or, in the absence of flow reversal, the so-called partially parabolic RANS equations, however, this simplification will not be considered here).

There are two possible approaches to the solution of the RANS equations: a global approach, in which one set of governing equations appropriate for both the inviscidand viscous-flow regions are solved using a large solution domain so as to capture the viscous-inviscid interaction; and an interactive approach, in which different sets of governing equations are used for each region and the complete solution obtained through the use of an interaction law, i.e., patching or matching conditions. Both approaches are depicted in Fig. 1. The former approach is somewhat more rigorous because it does not rely on the patching conditions that usually involve approximations. Nonetheless, for a variety of reasons, both types of approaches are of



FIG. 1. Definition sketch of flow-field regions and solution domains: (a) (x, y) plane; (b) (y, z) plane.

interest. In [15], both approaches were evaluated for zero Fr by comparing interactive and large-domain solutions for axisymmetric and simple three-dimensional bodies using the same numerical techniques and algorithms and turbulence model. It is shown that both approaches yield satisfactory results, although the interaction solutions appear to be computationally more efficient. As mentioned earlier, the present study utilizes the interactive approach. This takes advantage of the latest developments in both the inviscidand viscous-flow technologies; however, a large-domain

solution for the present problem is also of interest and a comparative evaluation as was done previously for zero Fr is planned for study under the present project for nonzero Fr.

Viscous–Inviscid Interaction

Referring to Fig. 1, there are two primary differences between the interactive and large-domain approaches with regard to the solution of the RANS equations: (1) the size

35

of the solution domain, i.e., the placement of the outer boundary S_o ; and (2) the boundary (i.e., edge) conditions specified thereon. For the large-domain solution, uniformflow and wave-radiation conditions are appropriate, whereas the interaction solution requires the specification of the match boundary (i.e., S_o), as well as an interaction law, and also a method for calculating the inviscid flow.

In the present study, solutions were obtained with the match boundary at about 2δ , where δ is the boundary layer and wake thickness. The interaction law is based on the concept of displacement thickness δ^* . A three-dimensional δ^* for a thick boundary layer and wake can be defined unambiguously by the two requirements that it be a stream surface of the inviscid flow continued from outside the boundary layer and wake and that the inviscid-flow discharge between this surface and any stream surface exterior to the boundary layer and wake be equal to the actual discharge between the body and wake centerplane and the latter stream surface. A method for implementing this definition for practical geometries is presently under development [17]; however, in lieu of this, an approximate definition is used in which two-dimensional definitions for δ^* , i.e.,

$$\delta^* = \int_0^\delta \left(1 - \frac{U}{U_p} \right) dr \tag{1}$$

for the keelplane and waterplane at each station are connected by a second-order polynomial.

In summary, the inviscid-flow solution is obtained for the displacement body δ^* . This solution then provides the boundary conditions for the viscous-flow solution, i.e.,

$$U(S_o) = U_p(S_o) = U_e,$$

$$W(S_o) = W_p(S_o) = W_e,$$

$$p(S_o) = p_p(S_o) = p_e.$$
(2)

Because δ^* and $V_p(S_o)$ are not known a priori, an initial guess must be provided and the complete solution obtained by iteratively updating the viscous- and inviscid-flow solutions until the patching conditions (1) and (2) are satisfied.

Viscous Flow

The viscous flow is calculated using the large-domain method of Patel *et al.* [14] modified and extended for interactive calculations and to include free-surface boundary conditions. The details of the basic method are provided by [14]. Herein, an overview is given as an aid in understanding the present modifications and extensions.

Equations and Coordinate System

The RANS equations are written in the physical domain using cylindrical coordinates (x, r, θ) as

$$\frac{\partial U}{\partial x} + \frac{1}{r} \frac{\partial}{\partial r} (rV) + \frac{1}{r} \frac{\partial W}{\partial \theta} = 0, \qquad (3)$$

$$\frac{DU}{Dt} = -\frac{\partial}{\partial x} \left(\hat{p} + \overline{uu} \right) - \frac{\partial}{\partial r} \left(\overline{uv} \right) - \frac{1}{r} \frac{\partial}{\partial \theta} \left(\overline{uw} \right) \\
- \frac{\overline{uv}}{r} + \frac{1}{\text{Re}} \nabla^2 U,$$
(4)

$$\frac{DV}{Dt} - \frac{W^2}{r} = -\frac{\partial}{\partial x} (\overline{uv}) - \frac{\partial}{\partial r} (\hat{p} + \overline{vv}) -\frac{1}{r} \frac{\partial}{\partial \theta} (\overline{vw}) - \frac{1}{r} (\overline{vv} - \overline{ww}) + \frac{1}{\text{Re}} \left(\nabla^2 V - \frac{2}{r^2} \frac{\partial W}{\partial \theta} - \frac{V}{r^2} \right),$$
(5)

$$\frac{DW}{Dt} + \frac{VW}{r} = -\frac{\partial}{\partial x} \left(\overline{uw}\right) - \frac{\partial}{\partial r} \left(\overline{vw}\right) - \frac{1}{r} \frac{\partial}{\partial \theta} \left(\hat{p} + \overline{ww}\right) \\ -\frac{2}{r} \left(\overline{vw}\right) + \frac{1}{\text{Re}} \left(\nabla^2 W + \frac{2}{r^2} \frac{\partial V}{\partial \theta} - \frac{W}{r^2}\right), \quad (6)$$

with

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + U \frac{\partial}{\partial x} + V \frac{\partial}{\partial r} + \frac{W}{r} \frac{\partial}{\partial \theta},$$

and

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial r^2} + \frac{1}{r}\frac{\partial}{\partial r} + \frac{1}{r^2}\frac{\partial^2}{\partial \theta^2}.$$

Closure of the RANS equations is attained through the use of the standard $k - \varepsilon$ turbulence model without modifications for free-surface effects. The limited experimental data available for surface-piercing bodies [18] indicate that, near a free surface, the normal component of turbulence is damped and the longitudinal and transverse components are increased. This effect has also been observed in open-channel flow [19] and in recent measurements for free-surface effects on the wake of a submerged flat plate [20] and a plane jet [21]. Such a turbulence structure cannot, in fact, be simulated with an isotropic eddy viscosity turbulence model like the present one; however, this aspect of the problem is also deferred for later study.

In the standard $k - \varepsilon$ turbulence model, each Reynolds
stress is related to the corresponding mean rate of strain by the isotropic eddy viscosity v_t as

$$-\overline{uv} = v_t \left(\frac{\partial U}{\partial r} + \frac{\partial V}{\partial x}\right),$$

$$-\overline{uw} = v_t \left(\frac{1}{r} \frac{\partial U}{\partial \theta} + \frac{\partial W}{\partial x}\right),$$

$$-\overline{vw} = v_t \left(\frac{1}{r} \frac{\partial V}{\partial \theta} + \frac{\partial W}{\partial r} - \frac{W}{r}\right),$$

$$-\overline{uu} = v_t \left(2 \frac{\partial U}{\partial x}\right) - \frac{2}{3}k,$$

$$-\overline{vv} = v_t \left(2 \frac{\partial V}{\partial r}\right) - \frac{2}{3}k,$$

$$-\overline{ww} = v_t \left(\frac{2}{r} \frac{\partial W}{\partial \theta} + 2 \frac{V}{r}\right) - \frac{2}{3}k;$$
(7)

 v_t is defined in terms of the turbulent kinetic energy k and its rate of dissipation ε by

$$v_t = C_\mu \frac{k^2}{\varepsilon},\tag{8}$$

where C_{μ} is a model constant and k and ε are governed by the modeled transport equations

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x} \left(\frac{1}{R_k} \frac{\partial k}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(\frac{1}{R_k} r \frac{\partial k}{\partial r} \right) + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{R_k} \frac{\partial k}{\partial \theta} \right) + G - \varepsilon,$$
(9)

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x} \left(\frac{1}{R_{\varepsilon}} \frac{\partial \varepsilon}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(\frac{1}{R_{\varepsilon}} r \frac{\partial \varepsilon}{\partial r} \right) \\
+ \frac{1}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{R_{\varepsilon}} \frac{\partial \varepsilon}{\partial \theta} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} G - C_{\varepsilon 2} \frac{\varepsilon^2}{k}.$$
(10)

G is the turbulence generation term,

$$G = v_t \left\{ 2 \left[\left(\frac{\partial U}{\partial x} \right)^2 + \left(\frac{\partial V}{\partial r} \right)^2 + \left(\frac{1}{r} \frac{\partial W}{\partial \theta} + \frac{V}{r} \right)^2 \right] + \left(\frac{\partial U}{\partial r} + \frac{\partial V}{\partial x} \right)^2 + \left(\frac{1}{r} \frac{\partial U}{\partial \theta} + \frac{\partial W}{\partial x} \right)^2 + \left(\frac{1}{r} \frac{\partial V}{\partial \theta} + \frac{\partial W}{\partial r} - \frac{W}{r} \right)^2 \right\}.$$
 (11)

The effective Re R_{ϕ} is defined as

$$\frac{1}{R_{\phi}} = \frac{1}{\text{Re}} + \frac{v_{\iota}}{\sigma_{\phi}}$$
(12)

in which $\phi = k$ for the k-equation (9) and $\phi = \varepsilon$ for the ε -equation (10). The model constants are: $C_{\mu} = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_U = \sigma_V = \sigma_W = \sigma_k = 1$, $\sigma_{\varepsilon} = 1.3$.

The governing equations (3) through (12) are transformed into nonorthogonal curvilinear coordinates such that the computational domain forms a simple rectangular parallelepiped with equal grid spacing. The transformation is a partial one since it involves the coordinates only and not the velocity components (U, V, W). The transformation is accomplished through use of the expression for the divergence and "chain-rule" definitions of the gradient and Laplacian operators which relate the orthogonal curvilinear coordinates $x^i = (x, r, \theta)$ to the nonorthogonal curvilinear coordinates $\xi^i = (\xi, \eta, \zeta)$. In this manner, the governing equations (3) through (12) can be rewritten in the form of the continuity and convective-transport equations

$$\frac{\partial}{\partial\xi} (b_1^1 U + b_2^1 V + b_3^1 W) + \frac{\partial}{\partial\eta} (b_1^2 U + b_2^2 V + b_3^2 W) + \frac{\partial}{\partial\zeta} (b_1^3 U + b_2^3 V + b_3^3 W) = 0,$$
(13)

$$g^{11} \frac{\partial^2 \phi}{\partial \xi^2} + g^{22} \frac{\partial^2 \phi}{\partial \eta^2} + g^{33} \frac{\partial^2 \phi}{\partial \zeta^2}$$
$$= 2A_{\phi} \frac{\partial \phi}{\partial \zeta} + 2B_{\phi} \frac{\partial \phi}{\partial \eta} + 2C_{\phi} \frac{\partial \phi}{\partial \xi} + R_{\phi} \frac{\partial \phi}{\partial t} + S_{\phi}. \quad (14)$$

Discretization and Velocity-Pressure Coupling

The convective-transport equations (14) are reduced to algebraic form through the use of a revised and simplified version of the finite-analytic method. In this method, Eqs. (14) are linearized in each local rectangular numerical element, $\Delta \xi = \Delta \eta = \Delta \zeta = 1$, by evaluating the coefficients and source functions at the interior node *P* and transformed again into a normalized form by a simple coordinate stretching. An analytic solution is derived by decomposing the normalized equation into one- and two-dimensional partial differential equations. The solution to the former is readily obtained. The solution to the latter is obtained by the method of separation of variables with specified boundary functions. As a result, a 12-point finite-analytic formula for unsteady, three-dimensional, elliptic equations is obtained in the form

$$\phi_{P} = \frac{1}{1 + C_{P}[C_{U} + C_{D} + (R/\tau)]}$$
(15)

$$\times \left\{ \sum_{1}^{8} C_{nb} \phi_{nb} + C_{P} \left(C_{U} \phi_{U} + C_{D} \phi_{D} + \frac{R}{\tau} \phi_{P}^{n-1} - S \right) \right\}.$$

It is seen that ϕ_P depends on all eight neighboring nodal values in the crossplane as well as the values at the upstream

and downstream nodes ϕ_U and ϕ_D , and the values at the previous time step ϕ_P^{n-1} . For large values of the cell Re, Eq. (15) reduces to the partially parabolic formulation which was used previously in other applications. Since Eq. (15) are implicit, both in space and time, at the current crossplane of calculation, their assembly for all elements results in a set of simultaneous algebraic equations. If the pressure field is known, these equations can be solved by the method of lines. However, since the pressure field is unknown, it must be determined such that the continuity equation is also satisfied.

The coupling of the velocity and pressure fields is accomplished through the use of a two-step iterative procedure involving the continuity equation based on the SIMPLER algorithm. In the first step, the solution to the momentum equations for a guessed pressure field is corrected at each crossplane such that continuity is satisfied. However, in general, the corrected velocities are no longer a consistent solution to the momentum equations for the guessed \hat{p} . Thus, the pressure field must also be corrected. In the second step, the pressure field is updated again through the use of the continuity equation. This is done after a complete solution to the velocity field has been obtained for all crossplanes. Repeated global iterations are thus required in order to obtain a converged solution. The procedure is facilitated through the use of a staggered grid. Both the pressure-correction and pressure equations are derived in a similar manner by substituting Eq. (15) for (U, V, W)into the discretized form of the continuity equation (13) and representing the pressure-gradient terms by finite differences.

Solution Domain and Boundary Conditions

The solution domain is shown in Fig. 1. In terms of the notation of Fig. 1, the boundary conditions on each of the boundaries are as follows: On the inlet plane S_i , the initial conditions for ϕ are specified from simple flat-plate and the inviscid-flow solutions. On the body surface S_b , a two-point wall-function approach is used. On the symmetry plane S_k , the conditions imposed are $\partial(U, V, \hat{p}, k, \varepsilon)/\partial\theta = W = 0$. On the exit plane S_e , axial diffusion is negligible so that the exit conditions used are $\partial^2 \phi / \partial x^2 = 0$, and a zero-gradient condition is used for \hat{p} . On the outer boundary S_o , the edge conditions are specified according to (2), i.e., $(U, W, \hat{p}) = (U_e, W_e, \hat{p}_e)$ and $\partial(k, \varepsilon)/\partial r = 0$, where (U_e, W_e, \hat{p}_e) are obtained from the inviscid-flow solution evaluated at the match boundary S_o .

On the free-surface S_{η} (or simply η), there are two boundary conditions, i.e.,

$$\mathbf{V} \cdot \mathbf{n} = 0 \tag{16}$$

and

$$\tau_{ij}n_j = \tau^*_{ij}n_j, \tag{17}$$

where **n** is the unit normal vector to the free surface and τ_{ij} and τ_{ij}^* are the fluid- and external-stress tensors, respectively, the latter, for convenience, including surface tension. The kinematic boundary condition expresses the requirement that η is a stream surface and the dynamic boundary condition that the normal and tangential stresses are continuous across it. Note that η itself is unknown and must be determined as part of the solution. In addition, boundary conditions are required for the turbulence parameters, k and ε ; however, at present, these are not well established.

In the present study, the following approximations were made in employing (16) and (17): (a) the external stress and surface tension were neglected; (b) the normal viscous stress and both the normal and tangential Reynolds stresses were neglected; (c) the curvature of the free surface was assumed small and the tangential gradients of the normal velocity components were neglected in the tangential stresses; and (d) the wave elevation was assumed small such that both (16) and (17) were represented by first-order Taylor series expansions about the mean wave-elevation surface (i.e., the waterplane S_w). Subject to these approximations, (16) and (17) reduce to

$$(U_x\eta_x + V_y\eta_y - W_z)|_{S_w} = 0, (18)$$

$$\hat{p}(S_w) = \eta / \mathrm{Fr}^2 - \eta \left. \frac{\partial \hat{p}}{\partial z} \right|_{S_w}, \quad (19)$$

$$\frac{\partial(\mathbf{V}, k, \varepsilon)}{\partial \theta}\Big|_{S_{w}} = 0, \tag{20}$$

where Cartesian coordinates (x, y, z) have been used in (18) and (19). Conditions (18) through (20) were implemented numerically as follows: The kinematic condition (18) was used to solve for the unknown free-surface elevation η by expressing the derivatives in finite-difference form and η in terms of its difference from an assumed (or previous) value. A backward difference was used for the x-derivative, a central difference for the y-derivative, and the inviscid-flow η_p was used as an initial guess. The dynamic conditions, (19) and (20), were used in conjunction with the solution for η in solving the pressure and momentum and turbulence model equations, respectively. Backward differences were used for the z- and θ -derivatives.

Inviscid Flow

The inviscid flow is calculated using the method of Rosen [2], i.e., the SPLASH computer code. The method is an extended version of the basic panel method of Maskew [22, 23] originally developed for the prediction of subsonic aerodynamic flows about arbitrary configurations, modified to include the presence of a free surface and gravity waves both for submerged and surface-piercing bodies. As is the

case with the basic method, lifting surfaces and their associated wake treatments as well as wall boundaries are included; however, the present overview and calculations are for nonlifting unbounded flow (see [24] for SPLASH results for lifting flow). The details of the basic method are provided by [22, 23]. Herein, an overview is given as an aid in understanding the extensions for the inclusion of the free surface and gravity waves and the present interaction calculations.

The flow is assumed irrotational such that the governing differential equation is the Laplace equation

$$\nabla^2 \phi = 0, \tag{21}$$

where ϕ is the external perturbation velocity potential; i.e.,

$$\mathbf{V}_p = U_o x + \nabla \phi. \tag{22}$$

A solution for ϕ may be obtained by defining also an internal perturbation potential ϕ_1 and applying Green's theorem to both the inner and outer regions and combining the resulting expressions to obtain

$$\phi = -\int_{S_b} \left\{ \mu \frac{\partial}{\partial n_Q} \left(\frac{1}{R_{PQ}} \right) + \frac{\sigma}{R_{PQ}} \right\} dS, \qquad (23)$$

where R_{PQ} is the distance from the surface point Q to the field point P and $\mu = \phi_1 - \phi$ and $\sigma = \partial(\phi - \phi_1)/\partial n_Q$ are the dipole and source strengths, respectively. In [22], the nature of solutions to (23) is investigated for two different specifications for ϕ_1 , i.e., $\phi_1 = 0$ and $U_o x$. In both cases, (23) is solved for the surface potential (i.e., $\phi(S_b)$) by representing the body by flat quadrilateral panels over which μ and σ are assumed constant and utilizing the farfield $\phi \to 0$ and body $\partial \phi/\partial n = -U_o n_x$ boundary conditions. The zero internal perturbation potential formulation ($\phi_1 = 0$) is shown to produce "results of comparable accuracy to those from higher-order methods for the same density of control points." In this case, the velocity normal to the external surface V_n is

$$V_n = U_a n_x + \partial \phi / \partial n = U_a n_x + \sigma \tag{24}$$

and, the velocity tangent to the external surface V_{t} is

$$V_t = U_o t_x + \partial \phi / \partial t = U_o t_x - \partial \mu / \partial t, \qquad (25)$$

where t_x is the x-component of a tangent vector and t is arclength in a tangential direction. For solid surfaces, V_n is usually zero, but it may be a specified nonzero value to simulate body motion, boundary-layer growth, inflow and outflow, control-surface deflection, etc. Hence, in the basic method, (24) is used to evaluate the source strengths directly. The corresponding doublet strengths are then given by solution of the discretized form of (23). Values of V_t are subsequently computed using (25) with a central difference for the *t*-derivative. It should be recognized that the so-called zero internal perturbation formulation is, in fact, equivalent to methods based on Green's third formula applied directly to the external perturbation potential (e.g., [25]).

In the SPLASH code, the internal zero-peturbation boundary condition is satisfied not only inside the submerged portion of the configuration, but also on the "other side" of a finite portion of the free surface. Both are represented by source-doublet singularity panels and flow leakage from one side of the free-surface to the other, at the free-surface outer boundary, is assumed to be negligible. This assumption is valid if the outer boundary of the free surface is sufficiently far from the configuration, and if the wave disturbances are eliminated before reaching the free-surface outer boundary. In this case, the discretized form of (23) is

$$\phi_i = \sum_{S_b + S_w} A_{ij} \mu_j + \sum_{S_b + S_w} B_{ij} \sigma_j = 0.$$
 (26)

The free-surface shape is determined by representing the undisturbed free surface by panels, whereupon free-surface boundary conditions linearized with respect to zero Fr are imposed [26]. The zero Fr velocities, U_o , V_o , and W_o , are obtained by first considering all free-surface panels as solid and fixed (in contrast to a traditional approach which employs the double panel or image model). The nonzero Fr velocities are then expressed as small increments to those for zero Fr. The velocities tangent and normal to a free-surface panel are, respectively,

$$U_x \approx U_o + \Delta U, \qquad V_v \approx V_o + \Delta V,$$
 (27)

and

$$V_n = W_z \approx W_o + \varDelta W \approx \varDelta W, \tag{28}$$

since $W_o = 0$ for a free-surface panel. Through Bernoulli's equation, the pressure on free-surface panels is a function of local velocity and is approximated by retaining only first-order incremental velocity terms

$$\hat{p} = \frac{1}{2} \{ 1 - (U_x^2 + V_y^2 + W_z^2) \}$$

$$\approx \frac{1}{2} \{ 1 - (U_o^2 + V_o^2) \} - \{ U_o \Delta U + V_o \Delta V \}$$

$$\approx \frac{1}{2} \{ 1 - (U_o^2 + V_o^2) \}$$

$$- \{ U_o (U_x - U_o) + V_o (V_y - V_o) \}.$$
(29)

Free-surface boundary conditions are linearized in a similar manner, retaining only first-order incremental

velocity and surface-elevation terms. The kinematic freesurface boundary condition (18) is approximated by

$$W_{z} = V_{n} \approx U_{o} \eta_{x} + V_{o} \eta_{y} \approx (U_{o}^{2} + V_{o}^{2})^{1/2} \eta_{so}, \quad (30)$$

where the subscript s_o denotes differentiation along a zero Fr streamline. The dynamic free-surface boundary condition (19), after differentiation along s_o , and substituting for η_{so} from (30), becomes

$$\frac{\partial \hat{p}}{\partial s_o} \approx \frac{1}{\mathrm{Fr}^2} \frac{V_n}{\left(U_o^2 + V_o^2\right)^{1/2}}.$$
(31)

A five-point backward difference is used in the ξ and η directions and the free-surface grid metrics are used to compute the pressure gradient

$$\frac{\partial \hat{p}}{\partial s_o} = \frac{U_o \frac{\partial \hat{p}}{\partial x} + V_o \frac{\partial \hat{p}}{\partial y}}{(U_o^2 + V_o^2)^{1/2}} = \frac{U_o \left(\frac{\partial \hat{p}}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial \hat{p}}{\partial \eta} \frac{\partial \eta}{\partial x}\right) + V_o \left(\frac{\partial \hat{p}}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial \hat{p}}{\partial \eta} \frac{\partial \eta}{\partial y}\right)}{(U_o^2 + V_o^2)^{1/2}}.$$
 (32)

The pressure-gradient algorithm is structured to permit the use of any blocked free-surface grid arrangement. Also, using less than a five-point backward difference tends to dampen wave amplitudes. This wave-damping mechanism is employed on panels near the outer boundary of the finite free-surface model, so that wave disturbances are eliminated before reaching the free-surface outer boundary.

At this point, a sufficient number of linear dependencies have been established to permit the elimination of the unknown free-surface source strengths in (26), i.e., (24) relates source strength to panel normal velocity, (31) relates free-surface panel normal velocity to streamwise pressure gradient, (32) with backward differences relates streamwise pressure gradient to free-surface pressures, (29) relates freesurface pressure to free-surface panel tangential velocities, (25) relates panel tangential velocities to the local surface gradient of doublet strength, and central differences relate the local surface gradient of doublet strength to doublet strengths. Hence, free-surface source strengths can be expressed as a linear combination of free-surface doublet strengths, i.e.,

$$\sigma_j = a_j + \sum_{S_w} b_{jk} \mu_k.$$
(33)

Substituting for σ_i from (33) into (26) yields

$$\phi_{i} = \sum_{S_{b} + S_{w}} A_{ij} \mu_{j} + \sum_{S_{b}} B_{ij} \sigma_{j} + \sum_{S_{w}} B_{ij} \left(a_{j} + \sum_{S_{w}} b_{jk} \mu_{k} \right).$$
(34)

With free-surface source strengths eliminated and source strengths on the solid body evaluated directly, solution of (34) yields the corresponding doublet strengths. The freesurface source strengths are then given by (33), and (24) and (25) are used to compute the resulting velocities on both body and free-surface panels. Pressures on free-surface panels are given by (29). A similar linearized formula is used for pressures acting on body panels, and configuration forces and moments are obtained by panel pressure integration.

For interactive calculations, the SPLASH code calculates the inviscid free-surface flow about the equivalent displacement body resulting from the previous viscous calculation. For this purpose, the equivalent displacement body is treated as a solid fixed surface. The inviscid flow velocities required for the next viscous flow calculation, at off-body points on the viscous grid outer boundary S_o , are obtained using the computed source-doublet solution and velocity influence coefficients. A sub-panel velocity influence-coefficient algorithm was developed which utilizes a bilinear variation of source and doublet strength across each panel. The continuous variation of source and doublet strength on each panel, and across panel edges, enhances the accuracy of off-body velocity calculations at points close to any body and/or free-surface panels.

WIGLEY HULL GEOMETRY AND EXPERIMENTAL INFORMATION

The Wigley parabolic hull was selected for the initial calculations since the geometry is relatively simple and it has been used in many previous computational and experimental studies. In particular, it is one of the two hulls, the other being the Series 60 $C_B = 0.6$ ship model, selected by the Cooperative Experimental Program (CEP) of the Resistance and Flow Committee of the International Towing Tank Conference [27] for which extensive global (total, wave pattern, and viscous resistance, mean sinkage and trim, and wave profiles on the hull) and local (hull pressure and wall shear-stress distributions and velocity and turbulence fields) measurements were reported. It was for these same reasons that the Wigley hull was selected as the first test case of the basic viscous-flow method [14], including comparisons with some of the zero Fr data of the CEP. Herein, comparisons are made for zero Fr with this same data and for nonzero Fr with the appropriate data of the CEP. As will be shown later, the nonzero Fr data is not as complete or of the same quality as that for zero Fr, which was the motivation for a related experimental study for the Series 60 $C_B = 0.6$ ship model [28] for which calculations and comparisons are in progress. However, the comparisons are still useful in order to validate the present interactive approach and display the shortcomings of both the computations and experiments.



FIG. 2. Computational grid: (a) longitudinal plane; (b) body and wake crossplanes.

The coordinates of the Wigley hull are given by

$$y = \frac{B}{2} \left\{ 4x(1-x) \right\} \left\{ 1 - (z/d)^2 \right\},$$
 (35)

where B = 0.1 and d = 0.0625. Waterplane and typical crossplane views are shown in Fig. 2.

RESULTS

In the following, first, the computational grids (Figs. 2 and 3) and conditions are described. Then, some example results are presented and discussed for zero Fr, followed by those for nonzero Fr, including, wherever possible, comparisons with available experimental data, and, in the latter case, with inviscid-flow results. The convergence history of the pressure is shown in Fig. 4. Figure 5 provides a comparison of the large-domain and interactive solutions. The free-surface perspective view and contours, wave profile, and surface-pressure profiles and contours are shown in Fig. 6 through 10, respectively. The axial-velocity contours, crossplane-velocity vectors, and pressure, axial-vorticity, and turbulent kinetic energy contours for several representative stations are shown in Figs. 11 through 13. Lastly, the velocity, pressure, and turbulent kinetic energy profiles for similar stations are shown in Figs. 14 through 16. On the figures and in the discussions, the terminology "interactive" refers to results from both the interactive viscous and displacement-body inviscid solutions. When the distinction is not obvious it will be made. The terminology "inviscid" or "bare-body" refers to the noninteractive inviscid solution.

Computational Grids and Conditions

The viscous-flow computational grid was obtained using the technique of generating body-fitted coordinates through the solution of elliptic partial differential equations. Because of the simplicity of the present geometry, it is possible to specify the axial f^1 and circumferential f^3 control functions as, respectively, only functions of ξ and ζ ; however, in order to accurately satisfy the body-surface boundary condition and resolve the viscous flow, $f^2 = f^2(\xi, \eta, \zeta)$. Partial views of the grids used in the calculations are shown in Figs. 2a, b for a longitudinal plane and typical body and wake crossplanes, respectively. Initially, a large-domain grid was generated. Subsequently, a small-domain grid was obtained by simply deleting that portion of the large-domain grid that lay beyond about r > 0.2. The outer boundary for the small-domain grid is shown by the dashed line in Fig. 2. For the large-domain grid, the inlet, exit, and outer boundaries are located at x = (0.296, 4.524) and r = 1, respectively. The first grid point off the body surface is located in the range $90 < y^+ < 250$. Fifty axial, 30 radial, and 15 circumferential grid points were used. As already indicated, the smalldomain grid was similar, except 21 radial grid points were used. In summary, the total number of grid points for the large- and small-domain calculations are 22,500 and 15,150, respectively.

The inviscid-flow displacement-body and free-surface panelization is shown in Fig. 3. Four hundred twenty three panels are distributed over the displacement body and 546 over the free surface for a total number of 969 panels. The panelization covers an area corresponding to 1-ship length upstream of the bow, 1.5-ship lengths in the transverse direction, and 3-ship lengths downstream of the stern. This panel arrangement was judged optimum based on panelization dependency tests [16].



FIG. 3. Displacement bodies: (a) Fr = 0; (b) Fr = 0.316.

The conditions for the calculations are as follows: L = 1; $U_0 = 1$; Re = 4.5 × 10⁶; Fr = 0 and 0.316; and on the inlet plane the average values for δ and U_{τ} are 0.0033 and 0.0455, respectively. These conditions were selected to correspond as closely as possible to those of the experiments of the CEP with which comparisons will be made [5, 29, 30].

Initially, large-domain calculations were performed for zero Fr. A zero-pressure initial condition was used and the values for the time α_t , pressure α_p , and transport quantity α_{ϕ} (where $\phi = k$ and ε) underrelaxation factors and total number of global iterations were 0.05 and 200, respectively. Next, small-domain calculations were performed, first for zero Fr, and then for nonzero Fr. For zero Fr, the interaction calculations were started with a zero-pressure initial condition and free-stream edge conditions ($U_e = 1$, $W_e = p_e = 0$). After 200 global iterations, the edge conditions were updated using the latest values of displacement thickness. Subsequently, the edge conditions were updated every 200 global iterations until convergence was achieved, which took three updates. For nonzero Fr, the calculations were started with the zero Fr solution as the initial condition and with nonzero Fr edge conditions obtained utilizing the zero Fr displacement body. This solution converged in 200 global iterations. Most of the results to be presented are for this case; however, some limited results will be shown in which the nonzero Fr edge conditions were obtained using an updated nonzero Fr displacement body. The values for $\alpha_{i}, \alpha_{n}, \text{ and } \alpha_{\phi}$ (where $\phi = k$ and ε) used for the small-domain calculations were the same as those for the large-domain calculations; however, for nonzero Fr, in addition, a value of 0.01 was used for α_{ϕ} (where $\phi = U$) for grid nodes near the outer boundary. The $\partial \hat{p}/\partial z$ term in (19) was found to have a small influence and was neglected in many of the calculations; however, this may be due, in part, to the present grid resolution. The calculations were performed on the Naval Research Laboratory CRAY XMP-24 supercomputer. The



FIG. 4. Convergence history.

CPU time required for the calculations was about 17 min for 200 global iterations for the viscous-flow code and 1 min for the inviscid-flow code.

Extensive grid dependency and convergence checks were not carried out since these had been done previously both for the basic viscous-flow method $\lceil 14 \rceil$ and for other applications. However, some calculations were performed using both coarser and finer grids. These converged, respectively, more rapidly and slower than the present solution. Qualitatively the solutions were very similar to the present one, but with reduced and somewhat increased resolution, respectively. The convergence criterion was that the change in solution be less than about 0.05% for all variables. Usually the solutions were carried out at least 50 global iterations beyond meeting this criterion. Figure 4 provides the convergence history for the pressure and is typical of the results for all the variables. In Fig. 4, the abscissa is the global iteration number it and the ordinate is the residual R(it), which is defined as

$$R(it) = \sum_{i=1}^{imax} |p(i, it-1) - p(i, it)| / \sum_{i=1}^{imax} |p(i, it1)|, \quad (36)$$

where *i*, itl, and imax are the grid-point index and total number of iterations, and grid points, respectively. Referring to Fig. 4, global iterations 1-200 correspond to the final iterations of the zero Fr solution and global iterations 200-400 to those for the nonzero Fr solution.

Zero Fr

Figure 5 provides a comparison of the zero Fr largedomain and interactive solutions and experimental data. The two solutions are nearly identical and show good agreement with the data, which validates the present interactive approach. The agreement with the data for the largedomain case is, of course, not surprising since this was already established in [14] for a similar grid and conditions, i.e., the present zero Fr solution is essentially the same as that of [14]. Some additional aspects of the zero Fr solution are displayed in Figs. 11 through 16 for later comparison with the nonzero Fr solution. Reference [14] provides detailed discussion of the zero Fr solution, including comparisons with the available experimental data. In summary, there is a downward flow on the forebody and an upward flow on the afterbody in response to the externalflow pressure gradients. The boundary layer and wake remain thin and attached and the viscous-inviscid interaction is weak; however, on the forebody, the boundary layer is relatively thicker near the keel than the waterplane, whereas the reverse holds true on the afterbody and in the near wake. The stern vortex is very weak. In the intermediate and far wake, the flow becomes axisymmetric.



FIG. 5. Comparison of interactive and large-domain solutions, waterplane: (a) surface and wake centerplane pressure; (b) wall-shear velocity; (c) wake centerplane velocity.

As indicated in Figs. 5 and 14 through 16, the agreement between the calculations and data is quite good; however, there are some important differences, which are primarily attributed to the deficiencies of the standard $k - \varepsilon$ turbulence model with wall functions. In particular, the axial velocity and turbulent kinetic energy are overpredicted near the stern and there is a more rapid recovery in the wake.

Nonzero Fr

Figure 5 also includes nonzero Fr results for comparison. On the waterplane, the surface and wake centerplane pressure displays very dramatic differences, the wall-shear velocity shows similar trends, but with reduced magnitude, and the wake centerplane velocity indicates a faster recovery in the intermediate and far wake. As will be shown later, the first closely follows the wave profile, the second is due to an increase in boundary-layer thickness near the waterplane for the nonzero Fr case, and the third can be explained by the wave-induced pressure gradients. On the keel, all three of these quantities are nearly the same as for zero Fr.

The free-surface perspective views (Fig. 6) and contours (Fig. 7) vividly display the complex wave pattern consisting of both diverging and transverse wave systems. The bow and stern wave systems are seen to initiate with crests and the shoulder systems initiate with troughs, which conforms to the usual pattern described for this type of hull form. Very apparent is the reduced amplitude of the stern waves for the interactive as compared to the inviscid solution. Also, the diverging wave system is more pronounced and at a smaller angle with respect to the centerplane. Note that the axial and transverse wave-induced pressure gradients can be discerned from these figures, but with an appropriate phase shift, i.e., increasing and decreasing wave elevations imply, respectively, adverse and favorable gradients. The wave profile along the hull is shown in Fig. 8, which, in this case, includes experimental data for comparison. On the forebody, the two solutions are nearly identical and underpredict the amplitude of the bow-wave crest and the first trough. On the afterbody, the interactive solution indicates larger values than the inviscid solution, with the data in between the two. The wave profile for the nonzero Fr displacement body (Fig. 3b) is also shown in Fig. 8. The differences are minimal on the forebody, whereas, they are significant on the afterbody and depart from the data. It appears that the present simple definition (1) is insufficient for "wavy" displacement bodies.

The surface-pressure profiles (Fig. 9) show similar tendencies as just discussed with regard to the wave profile. On the forebody, the two solutions are nearly identical, but, in this case, in very close agreement with the data. The pressure on the forebody shown by the dashed line is that obtained from the inviscid displacement-body solution. On the afterbody, here again, the interactive solution indicates larger values than the inviscid solution, with the data in between the two. The wave-induced effects are seen to diminish with increasing depth and the agreement between the two solutions and the data on the afterbody shows improvement. The surface-pressure contours (Fig. 10) graphically display the differences between the two solutions and the data. Note that the axial and vertical surfacepressure gradients can be discerned from these figures, i.e.,



FIG. 6. Free-surface perspective view: (a) interactive; (b) inviscid.

increasing and decreasing pressure imply, respectively, adverse and favorable gradients. The larger wave elevation and pressure on the afterbody for the interactive solution results in the closed contours near the stern displayed in Fig. 10b. As already mentioned, the viscous-inviscid interaction is weak for the Wigley hull, which is the reason that the inviscid and viscous pressure distributions are quite similar. However, it appears that the interaction is greater for nonzero as compared to zero Fr.

Figures 11 through 13 show the detailed results for several representative stations, i.e., x = 0.506, 0.904, and 1.112, although the discussion to follow is based on the complete results at all stations. Note that for zero Fr the upper boundary shown is the waterplane, whereas for non-



FIG. 7. Free-surface contours: (a) interactive; (b) inviscid.

zero Fr, it is the predicted free surface. Also, the axialvelocity, -vorticity, and turbulent kinetic energy contours are not shown for the inviscid solution since, in the former case, their values are all very close to one and, in the latter two cases, they are, of course, zero. Solid curves indicate clockwise vorticity.

On the forebody (Fig. 11), the boundary layer is thin such that many aspects of the solutions are similar; however, there are some important differences. The nonzero Fr pressure fields show local and global effects of the free surface; i.e., near the free surface, regions of high and low pressure coincide with wave crests and troughs, respectively, and at larger depths, the contours are parallel to the free surface. Also, for nonzero Fr, the crossplane-velocity vectors are considerably larger, especially for the interactive solution. The inviscid solution clearly lacks detail near the hull surface. The extent of the axial vorticity is increased for nonzero Fr and is locally influenced by the free surface. In both cases, as expected, the direction of rotation is mostly anticlockwise.

On the afterbody (Fig. 12), almost all aspects of the solutions show significant differences. The boundary layer



FIG. 8. Wave profile.



is thicker near the waterplane for nonzero as compared to zero Fr. This behavior begins at $x \approx 0.825$, which coincides with a region of adverse axial wave-induced pressure gradient (see Fig. 7). The differences for the pressure field and axial-vorticity contours are similar as described for the forebody; however, in the case of the crossplane-velocity vectors, there is an additional difference that, near the free surface, the interactive solution displays downward flow. This is consistent with the fact that the free-surface elevation is above the waterplane and the pressure is generally higher near the free surface than it is in larger depths, i.e., $\eta > 0$ and $\partial \hat{p}/\partial z < 0$. Note that, as expected, in both cases, the direction of rotation for the axial-vorticity is mostly clockwise. The turbulent kinetic energy contours are nearly the same for both Fr.

In the wake (Fig. 13), the solutions continue to show significant differences. Initially, the low-velocity region diffuses somewhat and covers a larger depthwise region; then, for x > 1.2, it recovers quite rapidly. A similar behavior was noted earlier for the wake centerline velocity for x > 1.2, both of which, as already mentioned, are consistent with the wave pattern. The zero Fr pressure field is nearly axisym-



FIG. 10. Surface-pressure contours: (a) experiment; (b) interactive; (c) inviscid.

metric and fully recovered by the exit plane. The nonzero Fr pressure field continues to show free-surface effects, i.e., the contours are parallel to the free surface, but also fully recovered by the exit plane. Note the considerably larger wave elevation near the wake centerplane for the inviscid as compared to the interactive solution, which was pointed out earlier with regard to Figs. 6 and 7. Here again, the crossplane-velocity vectors are larger for nonzero as compared to zero Fr, especially near the wake centerplane for the interactive solution. The interactive and inviscid solutions display differences near the free surface, which appear to be consistent with the differences in their predicted wave patterns. The zero Fr axial vorticity decays fairly rapidly, whereas, for nonzero Fr, the decay is slow with a layer of nonzero vorticity persisting near the free surface all the way to the exit plane. The turbulent kinetic energy contours are similar for both Fr, but recover faster for the nonzero case.

Figures 14 through 16 show the velocity, pressure, and



FIG. 11. Comparison of solutions at x = 0.506: (a) axial-velocity contours; (b) pressure contours; (c) crossplane-velocity vectors; (d) axial-vorticity contours; and (e) turbulent kinetic energy contours; columns, interactive, Fr = 0 and 0.316 and inviscid, Fr = 0.316, respectively.



FIG. 12. Comparison of solutions at x = 0.904: (a) axial-velocity contours; (b) pressure contours; (c) crossplane-velocity vectors; (d) axial-vorticity contours; and (e) turbulent kinetic energy contours; columns, interactive, Fr = 0 and 0.316 and inviscid, Fr = 0.316, respectively.



FIG. 13. Comparison of solutions at x = 1.112: (a) axial-velocity contours; (b) pressure contours; (c) crossplane-velocity vectors; (d) axial-vorticity contours; and (e) turbulent kinetic energy contours; columns, interactive, Fr = 0 and 0.316 and inviscid, Fr = 0.316.

SHIP BOUNDARY LAYERS AND WAKES



FIG. 14. Velocity, pressure, and turbulent kinetic energy profiles at x = 0.5.

turbulent kinetic energy profiles for similar stations as for Fig. 11 through 13, i.e., x = 0.5, 0.9, and 1.1. Also, included are both zero and nonzero Fr experimental data. At the largest two depths, z = 0.05 and 0.0625, data for both Fr are available, whereas, at the waterplane, z = 0, only zero Fr data are available. At the intermediate depths, data are available for both Fr, but for different z values. Since the interest here is primarily nonzero Fr and the zero Fr data

and comparisons were already displayed in [14], only nonzero Fr data are shown for z = 0.0125, 0.025, and 0.0375. For zero Fr, a corrected pressure is also shown which includes a constant (= -0.03) reference-pressure correction as described in [14]. Turbulent kinetic energy data are only available for zero Fr.

At x = 0.5, consistent with previous discussions, the differences between the two solutions are quite small



FIG. 15. Velocity, pressure, and turbulent kinetic energy profiles at x = 0.9.

and the agreemnt with the zero Fr data is good. However, the nonzero Fr data show some unexpected differences. In particular, the axial-velocity profile has a laminar appearance and the boundary-layer thickness is relatively large; the vertical velocity is upward, and the pressure shows considerable scatter. It is pointed out in [5] that the pressure-measurement error was appreciable.

At x = 0.9 and 1.1, here again, consistent with previous

discussions, the differences between the two solutions are significant and the agreement between the zero Fr solution and data is good, except for the aforementioned discrepancies. The nonzero Fr solution shows larger axial velocities than the measurements for the inner part of the profiles. Here again, the measured profiles have a laminar appearance and the boundary layer is thick. However, no doubt a part of the difference is due to the calculations; i.e., as is the

SHIP BOUNDARY LAYERS AND WAKES



FIG. 16. Velocity, pressure, and turbulent kinetic energy profiles at x = 1.1.

case for zero Fr, due to deficiencies of the $k - \varepsilon$ turbulence model, an overprediction of the velocity near the wall and wake centerplane is expected. The transverse velocity is small and with similar trends for both calculations and measurements. The calculations indicate downward vertical velocities near the free surface and upward values for the midgirth region and near the keel. The agreement with the data near the keel is satisfactory, but in the midgirth region and near the free surface the data display greater upward flow than the calculations. In the wake, the nonzero Fr data show surprisingly small vertical velocities near the wake centerplane. Here again, the nonzero Fr pressure data shows considerable scatter and is difficult to compare with the calculations. Consistent with earlier discussions the turbulent kinetic energy profiles are nearly the same for both Fr.

		Residuar	y-Resistanc	e Coefficie	ents	
	<i>L</i> (m)	T(°C)	$U_o ({ m m/s})$	Fr	Re	C_R
Experiment IHI	6	12.8	2.423	0.316	11.9 × 10 ⁶	1.803×10^{-3}
Experiment SRI	4	10.6	1.978	0.316	6.14	1.998
Experiment UT	2.5	17.3	1.564	0.316	3.6	1.866
Inviscid				0.316		1.79
Interactive	·			0.316	4.5×10^6	1.92

Lastly, Table I provides a comparison of the calculated pressure-resistance coefficient and experimental values of the residuary-resistance (i.e., total frictional) coefficient. The experimental values cover a range of Re, including the present value, and clearly show a dependency on Re. Interestingly, the inviscid result compares well with the data at the highest Re, whereas the interactive result is close to that which the data implies at the present Re.

WAVE-BOUNDARY LAYER AND WAKE INTERACTION

The comparisons of the zero and nonzero Fr interactive and inviscid-flow results with experimental data enables an evaluation of the wave-boundary layer and wake interaction. Very significant differences are observed between the zero and nonzero Fr interactive results due to the presence of the free surface and gravity waves. In fact, the flow field is completely altered. Most of the differences were explicable in terms of the differences between the zero and nonzero Fr surface-pressure distributions and, in the latter case, the additional pressure gradients at the free surface associated with the wave pattern. The viscous-inviscid interaction appears to be greater for nonzero as compared to zero Fr. It should be mentioned that other factors undoubtedly have important influences, e.g., wave-induced separation, which are not included in the present theory.

The interactive and inviscid nonzero Fr solutions also indicate very significant differences. The inviscid solution clearly lacks "real-fluid effects." The viscous flow close to the hull and wake centerplane is clearly not accurately resolved. The interactive solution shows an increased response to pressure gradients as compared to the inviscid solution, especially in regions of low velocity. Also, the inviscid solution overpredicts the pressure recovery as the stern and the stern-wave amplitudes.

CONCLUDING REMARKS

The present work demonstrates for the first time the feasibility of an interactive approach for calculating ship boundary layers and wakes for nonzero Fr. The results presented for the Wigley hull are very encouraging. In fact,

in many respects, the present results appear to be superior to the only other solutions of this type available, i.e., [10, 11]. This is true both with regard to the resolution of the boundary-layer and wake regions and the wave field. Furthermore, it appears that the present interactive approach is considerably more computationally efficient than the large-domain approaches of [10, 11]. This is consistent with the previous finding for zero Fr [15]. However, a complete evaluation of the present method was not possible. In the former case, due to the limited available experimental data. As mentioned earlier, a related experimental study for the Series 60 $C_B = 0.6$ ship model [28] was recently completed for which extensive measurements were made at both low and high Fr for which calculations and comparisons are in progress. In the latter case, due to the considerable differences in numerical techniques and algorithms and turbulence models between the present methods and those of [10, 11]. As mentioned earlier, the pursuit of a large-domain approach to the present problem is also of interest and will enable such an evaluation.

Finally, some of the issues that need to be addressed while further developing and validating the present approach are as follows: further assessment of the most appropriate free-surface boundary conditions; improved definition and construction of displacement bodies; the inclusion and resolution of the bow-flow region; extensions for lifting flow; and the ever present problem of grid generation and turbulence modeling. Also, of interest is the inclusion of nonlinear effects in the inviscid-flow code.

APPENDIX: NOMENCLATURE

4 D 4	Coefficients in transport equations
$A_{\phi}, B_{\phi},$ etc.	Coefficients in transport equations
$A_{ii}, B_{ii}, a_i, b_{jk}$	Influence coefficients
b_i^i	Geometric tensor
$\dot{C}_D, C_P, C_U, C_{\rm nb}$	Finite-analytic coefficients ($nb = NE$, NW , SE , etc.)
C_{f}	Friction coefficient $(=2\tau_w/\rho U_o^2)$
Ć,	Pressure coefficient
C_{R}^{\prime}	Residuary-resistance coefficient $(=2R/\rho SU_o^2)$
Fr	Froude number $(=U_o/\sqrt{gL})$
g ^{ij}	Conjugate metric tensor in general curvilinear coor-
0	dinates ξ^i
k	Turbulent kinetic energy

SHIP BOUNDARY LAYERS AND WAKES

L	Characteristic (ship) length			
n	Normal unit vector			
\hat{p}	Piezometric pressure			
R	Residuary resistance			
Re	Reynolds number $(=U_{a}L/v)$			
S	Wetted surface area			
S_b, S_e , etc.	Boundaries of the solution domain			
S_{ϕ}, S	Source functions			
t	Time; arclength in tangential direction			
t -	Tangent unit vector			
U, V, W	Velocity components in cylindrical polar coordinates			
U_x, V_y, W_z	Velocity components in Cartesian coordinates			
U_c	Wake centerline velocity			
U_o	Characteristic (freestream) velocity			
U_{τ}	Wall-shear velocity $(=\sqrt{\tau_w/\rho})$			
uu , vv, etc.	Reynolds stresses			
x, y, z	Cartesian coordinates			
x, r, θ	Cylindrical polar coordinates			
x^+, y^+, z^+	Dimensionless distances (= $U_{\tau} x/v$, etc.)			
δ^*	Displacement thickness			
3	Rate of turbulent energy dissipation			
η	Free-surface elevation			
μ	Dipole strength			
ν	Kinematic viscosity			
v _t	Eddy viscosity			
ξ, η, ζ	Body-fitted coordinates			
ρ	Density			
σ	Source strength			
τ	Time increment			
τ_{ij}, τ^*_{ij}	Fluid- and external-stress tensors			
τ_w	Wall-shear stress			
ϕ	Transport quantities (U, V, W, k, ε); velocity poten-			
	tial			

Subscripts

е	edge value
0	freestream or zero Fr value
р	inviscid-flow value

ACKNOWLEDGMENTS

This research was sponsored by the Office of Naval Research under Contract N00014-88-K-0113 under the administration of Dr. E. P. Rood whose support and helpful technical discussions are greatly appreciated.

REFERENCES

- 1. W. Lindenmuth, T. J. Ratcliffe, and A. M. Reed, DTRC/SHD-1260-1, 1988 (unpublished).
- 2. B. Rosen, South Bay Simulations Inc., Babylon, NY, private communication (1989).
- 3. V. C. Patel, in Proceedings, 17th Office of Naval Research Symposium on Naval Hydrodynamics, The Hague, The Netherlands, 1988, p. 217.

- 4. Proc. 5th International Conference on Numerical Ship Hydrodynamics, Hiroshima, Japan, 1989.
- 5. A. Shahshahan, Ph.D. thesis, The University of Iowa, Iowa City, IA, 1985 (unpublished).
- 6. M. Ikehata and Y. Tahara, J. Soc. Naval Architects Jpn. 161, 49 (1987) [Japanese].
- 7. F. Stern, J. Ship Res. 30, No. 4, 256 (1986).
- 8. F. Stern, W. S. Hwang, and S. Y. Jaw, J. Ship Res. 33, No. 1, 63 (1989).
- 9. F. Stern, in Proceedings, 4th International Conference on Numerical Ship Hydrodynamics, Washington, DC, 1985, p. 383.
- 10. H. Miyata, T. Sato, and N. Baba, J. Comput. Phys. 72, No. 2, 393 (1987).
- 11. T. Hino, in Proceedings, 5th International Conference on Numerical Ship Hydrodynamics, Hiroshima, Japan, 1989, p. 103.
- 12. F. H. Harlow and J. E. Welch, Phys. Fluids 8, 2182 (1965).
- 13. R. K. C. Chan and R. L. Street, J. Comput. Phys. 6, 68 (1970).
- 14. V. C. Patel, H. C. Chen, and S. Ju, Iowa Institute of Hydraulic Research, The University of Iowa, IIHR Report No. 323, 1988 (unpublished); J. Comput. Phys. 88, No. 2, 305 (1990).
- 15. F. Stern, S. Y. Yoo, and V. C. Patel, AIAA J. 26, No. 9, 1052 (1988).
- 16. J. Longo, M.S. thesis, The University of Iowa, Iowa City, IA, 1990 (unpublished).
- 17. R. Black, M.S. thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- 18. T. Hotta and S. Hatano, in Fall Meeting of the Society of Naval Architects of Japan, 1983.
- 19. W. Rodi, "Turbulence Model and Their Application in Hydraulics," presented at the IAHR Section on Fundamentals of Division II: Experimental and Mathematical Fluid Dynamics, 1980 (unpublished).
- 20. T. F. Swean and R. D. Peltzer, NRL Memo Report 5426, Naval Research Laboratory, Washington, DC, 1984 (unpublished).
- 21. S. E. Ramberg, T. F. Swean, and M. W. Plesniak, NRL Memo Report 6367, Naval Research Laboratory, Washington, DC, 1989 (unpublished).
- 22. B. Maskew, J. Aircraft 19, No. 2, 157 (1982).
- 23. B. Maskew, NASA CR-166476, 1982 (unpublished).
- 24. C. W. Boppe, B. S. Rosen, J. P. Laiosa, and B. Chance, Jr., "Stars & Stripes '87: Computational Flow Simulations for Hydrodynamic Design," the Eighth Chesapeake Sailing Yacht Symposium, Annapolis, MD, 1987 (unpublished).
- 25. F. Stern, ASME J. Fluids Eng. 111, 290 (1989).
- 26. C. W. Dawson, in Proceedings, 2nd International Conference on Numerical Ship Hydrodynamics, Berkeley, CA 1977, p. 30.
- 27. "Report of the Resistance and Flow Committee," in Proceedings, 18th Int. Towing Tank Conf., Kobe, Japan, 1987, p. 47.
- 28. Y. Toda, F. Stern, and J. Longo, Iowa Institute of Hydraulic Research, The University of Iowa, IIHR report, 1991 (unpublished).
- 29. O. P. Sarda, Ph.D. thesis, The University of Iowa, Iowa City, IA, 1986 (unpublished).
- 30. H. Kajatani, H. Miyata, M. Ikehata, H. Tanaka, H. Adachi, M. Namimatsu, and S. Ogiwara, in Proceedings, 2nd DTNSRDC Workshop on Ship Wave-Resistance Computations, 1983, p. 5.

Printed by Catherine Press, Ltd., Tempelhof 41, B-8000 Brugge, Belgium

53

Nonlinear Free-Surface Flow in Oscillating Channel during Earthquakes

By Yusuke Tahara¹ and Allen T. Chwang,² Fellow, ASCE

ABSTRACT: The nonlinear free-surface flow in an infinitely long channel during earthquakes is analyzed by the boundary element method. The solution is obtained by a distribution of simple Rankine-type singularities on the walls of the channel and on the undisturbed free surface. The fluid in the channel is assumed to be inviscid and incompressible, and its motion irrotational. Unsteady velocity potential is determined with second-order nonlinear boundary conditions applied on the free surface. The time-marching procedure is introduced to solve for the transient stage after an earthquake takes place. The channel is assumed to be rigid during an earthquake, and the amplitude of the channel oscillation and the free-surface elevation are assumed to be small such that the free-surface boundary conditions are represented by Taylor-series expansions about the mean water surface. An overview is given for the present approach, and numerical results are presented for twodimensional flows due to horizontal, harmonic ground accelerations, including comparisons between linear and nonlinear solutions.

INTRODUCTION

An important factor in the design of channels in seismic regions is the accurate determination of the critical free-surface elevation during earthquakes. This is a classical and important problem in civil engineering. Initially, the interest was primarily with simple geometries with linearized free-surface boundary conditions. Nowadays, the emphasis is placed on applications for more complicated geometries with nonlinear free-surface boundary conditions. The present study is central to the aforementioned problems, i.e., it concerns the development of numerical approach for calculating two-dimensional flows due to earthquakes.

Considerable effort has been put forth in the investigation of earthquakerelated problems. For a two-dimensional dam with a vertical upstream face, Westergaard (1933) first determined the hydrodynamic pressure on the vertical dam face due to horizontal harmonic ground motion in the direction perpendicular to the dam. Kotsubo (1959, 1961) obtained a general solution for both transient and steady-state hydrodynamic pressures acting on rigid concrete dams. Chwang (1981) analyzed the effect of stratification of the fluid in the reservoir on hydrodynamic pressures on dams.

For a dam with a nonvertical upstream face, Chwang and Housner (1978) solved analytically the two-dimensional problem of the added-mass effect due to a horizontal acceleration of a rigid dam with an inclined upstream face of constant slope by adopting the generalized von Kàrmàn momentum-balance approach. Chwang (1978) presented an integral solution for the earthquake force on a rigid, sloping dam based on the exact, two-dimensional potential-flow theory.

¹Postdoctoral Assoc., Dept. of Mech. Engrg., Inst. of Hydr. Res., 301A Hydr. Lab., Univ. of Iowa, Iowa City, IA 52242.

²Prof. and Head, Dept. of Mech. Engrg., Univ. of Hong Kong, Pokfulam Road, Hong Kong.

Note. Discussion open until September 1, 1993. To extend the closing date one month, a written request must be filed with the ASCE Manager of Journals. The manuscript for this paper was submitted for review and possible publication on February 4, 1992. This paper is part of the *Journal of Engineering Mechanics*, Vol. 119, No. 4, April, 1993. ©ASCE, ISSN 0733-9399/93/0004-0801/\$1.00 + \$.15 per page. Paper No. 3346.

On the other hand, Moiseev (1958) and Faltinsen (1974) performed studies of nonlinear vibrations of a finite volume of liquid. Chan and Chan (1980) treated the steady and transient free-surface flows about a ship with a blunt bow by using the finite difference numerical scheme. Chwang (1983) developed a nonlinear, small-time-expansion method to determine the freesurface profile and the impulsive force on a suddenly moving vertical plate with a horizontal acceleration. Following this concept, Chwang and Wang (1984) solved the nonlinear problem for an accelerating, rectangular, or circular container.

In these mentioned studies, the analytical or numerical solutions are valid only for simple geometries. This is mainly due to the nonlinearity of the problem, where there is a nonlinear free-surface boundary condition imposed at unknown boundary. Hence, combination of analytical and numerical techniques is very useful and effective for more general, complicated geometries with nonlinear free-surface effects.

In this paper, the proposed method is a boundary element method with Rankine-type sources as the kernel function. The fluid is assumed to be inviscid and incompressible, and its motion irrotational. Unsteady velocity potential is solved with the nonlinear free-surface boundary conditions derived by the second-order perturbation method. The time-marching procedure is introduced to solve for the transient stage after an earthquake takes place. The channel is assumed to be rigid during an earthquake, and the motion is due to horizontal, harmonic ground accelerations. The amplitude of the channel oscillation and the free-surface elevation are assumed to be small such that the free-surface boundary conditions are represented by Taylor-series expansions about the mean water surface.

GOVERNING EQUATIONS

Let us consider an infinitely long channel with uniform cross section as depicted in Fig. 1. The y-axis points upwards and the x-axis is perpendicular to the y-axis in the horizontal plane. The channel bottom is the y = 0 plane and the undisturbed water surface is at y = h. The walls are assumed to be rigid. An earthquake starts at t = 0; then the channel begins to oscillate with an acceleration $-a\omega^2 \sin \omega t$ in the x-direction for t > 0. The displacement and the velocity of the channel corresponding to this ground acceleration are $a \sin \omega t$ and $a\omega \cos \omega t$, respectively, in the x-direction. The maximum displacement of the channel, a, is assumed to be small, as is the deviation of the free surface from its undisturbed level, $\eta(x, t)$.

Since the fluid in the channel is assumed to be inviscid and incompressible and its motion irrotational, the velocity field can be defined as



FIG. 1. Uniform Cross Section of Infinitely Long Channel

where u and v = the velocity components in the x- and y-directions, respectively; and $\Phi(x, y, t) =$ the velocity potential that satisfies the Laplace equation

in the fluid domain D. The boundary condition on the channel wall S_0 is

$$\frac{\partial \Phi}{\partial n} = U_{0n} \qquad \dots \qquad \dots \qquad \dots \qquad (3)$$

where $\mathbf{n} = (n_x, n_y)$ denotes the unit inward normal vector on the boundary (see Fig. 1), and U_{0n} is given by

On the free surface $y = h + \eta(x, t)$, denoted as S_1 , the kinematic and dynamic boundary conditions are

$$\frac{\partial \eta}{\partial t} + \frac{\partial \Phi}{\partial x} \frac{\partial \eta}{\partial x} = \frac{\partial \Phi}{\partial y} \qquad (5a)$$

where $\mathbf{g} =$ the gravitational constant.

The present problem is nonlinear because the free-surface boundary conditions in (5a) and (5b) are nonlinear. Eqs. (5a) and (5b) should be satisfied on the unknown free surface, which is a part of the solution. In its exact form, the present problem is difficult to solve. We shall assume that η and Φ can be expanded in terms of a small parameter ε , say $\varepsilon = a/h$, as

Substituting (6a) and (6b) into (5b) and applying Taylor-series expansions about undisturbed free surface, we have ε^1 terms

$$\frac{\partial \Phi_1}{\partial t} + g\eta_1 = 0, \qquad y = h \qquad \dots \qquad (7a)$$

and ε^2 terms

$$\frac{\partial \Phi_2}{\partial t} + g\eta_2 = -\eta_1 \frac{\partial^2 \Phi_1}{\partial t \partial y} - \frac{1}{2} \left(\frac{\partial \Phi_1}{\partial x} \right)^2 - \frac{1}{2} \left(\frac{\partial \Phi_1}{\partial y} \right)^2, \qquad y = h \quad \dots \quad (7b)$$

Similarly, (6a), (6b), and (5a) yield ε^1 terms

$$\frac{\partial \eta_1}{\partial t} - \frac{\partial \Phi_1}{\partial y} = 0, \qquad y = h \qquad (8a)$$

and ε^2 terms

The solution of (2), satisfying boundary conditions in (3), (7*a*), and (8*a*), gives the first-order velocity potential Φ_1 . The solution of (2), satisfying (3), (7*b*), and (8*b*), gives the second-order velocity potential Φ_2 . In this study, Φ is obtained up to the second order.

NUMERICAL APPROACH

The velocity potential at field point P(x, y) induced by a singularity distribution on the boundaries is given by

$$\Phi(x, y, t) = \int_{S_0} \sigma(\ln r + \ln \bar{r}) \, ds + \int_{S_1} \sigma \ln r \, ds \quad \dots (9)$$

where r = the distance between field point P and a source point; $\bar{r} =$ the distance between P and an image of the source point about the undisturbed free surface; and $\sigma =$ the source density. All surfaces are divided into straight-line segments l_j , and the source densities σ_j are approximated by a constant value on each segment. Integration is replaced by summation. Thus, velocity potentials Φ_1 and Φ_2 are given by

where N = the total number of segments and C_j is given by

and

with

$$r = \sqrt{(x - x_j)^2 + (y - y_j)^2}$$
(11d)

A time-marching procedure is introduced here to obtain the unsteady velocity potentials. At the *n*th time step, the boundary condition in (3), which is satisfied by Φ_1 and Φ_2 at the central point of a line segment l_i , i.e, $P(x_i, y_i)$, is written as

$$\pi \sigma_{ki}^n + \sum_{j=1}^N \mathbf{A}_{ij} \sigma_{kj}^n = U_{0n}^n, \qquad k = 1, 2 \quad \dots \quad \dots \quad \dots \quad \dots \quad \dots \quad \dots \quad (12a)$$

where

$$\mathbf{A}_{ij} = n_{xi} \int_{l_j} \frac{\partial G(r_{ij})}{\partial x_i} \, ds_j + n_{yi} \int_{l_j} \frac{\partial G(r_{ij})}{\partial y_i} \, ds_j, \quad \text{for } i \neq j \quad \dots \quad (12b)$$

-47-

On the free surface, discretization of (7a) and (8a) yields

Similarly, from (7b) and (8b), we have

$$\Phi_2^n = \Phi_2^{n-1} - \left[g\eta_2^{n-1} + \eta_1^{n-1} \frac{\partial^2 \Phi_1^{n-1}}{\partial t \partial y} + \frac{1}{2} \left(\frac{\partial \Phi_1^{n-1}}{\partial x} \right)^2 + \frac{1}{2} \left(\frac{\partial \Phi_1^{n-1}}{\partial y} \right)^2 \right] \cdot \Delta t \qquad (14a)$$

$$\eta_{2}^{n+1} = \eta_{2}^{n} - \left[\pi \sigma_{2}^{n} + \frac{\partial \Phi_{1}^{n}}{\partial x} \frac{\partial \eta_{1}^{n}}{\partial x} - \eta_{1}^{n} \frac{\partial^{2} \Phi_{1}^{n}}{\partial y^{2}} \right] \cdot \Delta t$$
$$= \eta_{2}^{n} - \left[\pi \sigma_{2}^{n} + \frac{\partial \Phi_{1}^{n}}{\partial x} \frac{\partial \eta_{1}^{n}}{\partial x} + \eta_{1}^{n} \frac{\partial^{2} \Phi_{1}^{n}}{\partial x^{2}} \right] \cdot \Delta t \qquad (14b)$$

Substituting (10) and (11) into (13*a*) and (14*a*), we obtain matrix equations for unknown source densities σ_1 and σ_2 on the undisturbed free surface

$$\sum_{j=1}^{N} \mathbf{B}_{ij} \sigma_{1j}^{n} = \sum_{j=1}^{N} \mathbf{B}_{ij} \sigma_{1j}^{n-1} - \mathbf{g} \eta_{1i}^{n-1} \cdot \Delta t \qquad (15a)$$

$$\sum_{j=1}^{N} \mathbf{B}_{ij} \sigma_{2j}^{n} = \sum_{j=1}^{N} \mathbf{B}_{ij} \sigma_{2j}^{n-1} - \left[\mathbf{g} \eta_{2i}^{n-1} + \eta_{1i}^{n-1} \frac{\partial^2 \Phi_{1i}^{n-1}}{\partial t \partial y} + \frac{1}{2} \left(\frac{\partial \Phi_{1i}^{n-1}}{\partial x} \right)^2 + \frac{1}{2} \left(\frac{\partial \Phi_{1i}^{n-1}}{\partial y} \right)^2 \right] \cdot \Delta t \qquad (15b)$$

where

$$B_{ij} = \int_{l_j} G(r_{ij}) \, ds_j \qquad \dots \qquad (16)$$

Eqs. (12a)-(12d), which are valid on the channel wall S_0 , and (15a) and (15b) which are valid on the undisturbed free surface y = h, give the complete matrix equations for σ_1^n and σ_2^n . The matrices in (12a)-(12d) and (15a)-(15b) do not change in the time-marching procedure. After σ_1^n and σ_2^n are determined, the free-surface elevations η_1^{n+1} and η_2^{n+1} are obtained from (13b) and (14b), respectively.

The overall solution procedure is summarized as follows:

1. Approximate the undisturbed free surface and the channel wall enclosing the fluid domain by straight-line segments.

2. Calculate geometrical terms in the matrix equations.

3. Specify initial conditions, i.e., $\Phi_1 = \Phi_2 = \eta_1 = \eta_2 = 0$.

4. Specify boundary conditions at the present time step.

5. Solve matrix equations (12a) and (15a) for Φ_1^n .

6. Calculate η_1 by (13b).

7. Solve the matrix equations (12a) and (15b) for Φ_2^n .

8. Calculate η_2 by (14b).

9. Repeat steps 4-8 for all subsequent time steps until the desired time is reached.

RESULTS

In the following, the computational conditions of the numerical cases are described, followed by results of segment density tests that demonstate the accuracy of the numerical approach. Lastly, numerical results are presented and discussed for linear and nonlinear solutions. In this section and its figure, the term *linear* refers to the first-order solution and *nonlinear* to the second-order solution. The results of free-surface elevation are nondimensionalized using channel base width b.

Consider a channel with inclined walls of constant slope tan α , as shown in Fig. 1. Let $a = \varepsilon h$ be the maximum amplitude of channel displacement due to an earthquake in the x-direction, where $0 < \varepsilon << 1$. The ratio of the base width to the channel depth is b/h = 8.5, and the Froude number is $\omega \sqrt{h/g} = 22$. This Froude number gives relatively small values of the dimensional parameter $C = g/(\omega^2 h)$, which is a ratio of the gravity effect to the inertial effect due to oscillation (Chwang 1981). A small value of C means that the gravity effect is small, or the inertial effect due to oscillation is large.

The effect of segment density is examined using linear solutions for a channel of rectangular cross section with $\varepsilon = 0.001$. An earthquake is assumed to take place at t = 0, and the resultant flow motion will eventually become standing waves through a transient stage. In the computation, standing-wave solutions are obtained at no later than t = 10T, where T is the time period. The time increment Δt is set to be 0.01T, based on a numerical stability analysis. The analytical expressions of Φ_1 and η_1 for standing waves are

where

$$P_n = \sin k_n h \qquad (18d)$$

and k_0 and k_n = solutions of $1 - Ck_0h$ tanh $k_0h = 0$ and $1 + Ck_nh$ tan $k_nh = 0$, respectively.

In Fig. 2, the linear wave elevations at x/b = -0.45 are plotted versus the dimensionless time t/T for analytical and numerical solutions. The numerical solutions are obtained using total segment numbers of N = 200, 400, and 600. It appears that N = 400 gives sufficient accuracy and this segment density is used for later calculations.

The linear and nonlinear wave elevations at $x/b_w = 0.5$ are plotted versus t/T in Figs. 3(a) and 3(b), respectively, for tan $\alpha = 2.0$ and $\varepsilon = 0.001$, where b_w denotes the width of the mean water surface of the channel at y = h. Figs. 3(a) and 3(b) show that both linear and nonlinear solutions have already become standing-wave solutions at t/T = 10. Also, nonlinear solutions show slightly higher wave elevations than the linear solutions although both coincide at t = 0.

Figs. 4(a)-7(b) show typical wave profiles of linear and nonlinear standing-wave solutions for different values of ε and tan α . These figures show that free-surface waves for large values of tan α have large amplitudes, as expected based on physical intuition; the smaller the value of ε is, the smaller the differences between linear and nonlinear solutions would be. This is logical because the nonlinear effect diminishes as ε decreases. It is also seen that nonlinear solutions, and are not symmetric about the central point of the mean water level. However, the linear wave profiles do not coincide with the mean water level at t/T = 10 and t/T = 10.5, which is likely due to numerical errors.



FIG. 2. Linear Wave Elevations at x/b = -0.45 versus Dimensionless Time t/T



Waves

-10-3

۴

ŝ



¢



= 0.001 and tan α = 0.5 for (a) Linear Waves; and (b) Nonlinear

Waves

-52-







FIG. 9. Maximum Wave Elevation at Channel Wall versus Channel Slope tan α



FIG. 10. Maximum Wave Elevation at Channel Wall versus Dimensionless Channel Displacement ϵ

Fig. 5(b) shows that nonlinear wave profiles for $\varepsilon = 0.005$ and tan $\alpha = 2.0$ are quite different from the linear wave profiles. As the wall accelerates into the fluid, the fluid starts to pile up. However, when the wall accelerates in the reverse direction, the free surface does not recover even to the mean water level. This phenomenon is not present in the linear theory, which assumes that the mean water surface is a plane of symmetry. Figs. 8(a) and 8(b) provide enlarged pictures of the free surface near the wall region so as to show the nonlinear effect more closely. In both figures, wave profiles are extrapolated for $x/b_w > 0.5$ using cubic splines.

In Fig. 9, the maximum nonlinear wave elevation at the channel wall is plotted versus tan α for $\varepsilon = 0.001$ and 0.005. This is the highest wave elevation at the wall; a region in which the free-surface is creeping up the wall as shown in Fig. 8(b). As tan α tends to zero, the maximum value of η , η_m , tends to zero. The wave elevation for $\varepsilon = 0.005$ is much higher than that for $\varepsilon = 0.001$. As discussed, the nonlinear effect is very important in the accurate estimation of η_m . For a fixed channel slope of tan $\alpha = 2.0$, the maximum wave height η_m is plotted versus ε in Fig. 10. In Fig. 10, as ε increases, η_m increases rapidly.

CONCLUSIONS

The present work demonstrates the feasibility of a boundary element method for solving a nonlinear free-surface problem for two-dimensional channels during earthquakes. Results presented in this work are encouraging, and may be useful for estimating the critical free-surface elevation during earthquakes. The calculations have been done on a Cray-Y/MP supercomputer, and the present approach is computationally more efficient than other numerical approaches, such as the marker-and-cell (MAC)-type method. In fact, all the calculations are done within 30 s. It is planned to extend the present method to more complicated geometries.

ACKNOWLEDGMENT

The first writer wishes to thank the Weeg Computing Center of the University of Iowa for student allocation of supercomputing hours at the National Center for Supercomputing Applications.

APPENDIX. REFERENCES

- Chan, K. C., and Chan, W. K. (1980). "Numerical solution of transient and steady free-surface flows about a ship of general hull shape." *Proc. of 13th Symp. on Naval Hydrodynamics*, 257-276.
- Naval Hydrodynamics, 257-276. Chwang, A. T. (1978). "Hydrodynamic pressures on sloping dams during earthquakes. Part 2: Exact theory." J. Fluid Mech., 87(2), 343-347.
- Chwang, A. T. (1981). "Effect of stratification on hydrodynamic pressures on dams." J. Engrg. Math., 15(1), 49-63.
- Chwang, A. T. (1983). "Nonlinear hydrodynamic pressure on an accelerating plate." *Physics of Fluids*, 26(2), 383–387.
- Chwang, A. T., and Housner, G. W. (1978). "Hydrodynamic pressures on sloping dams during earthquakes. Part 1: Momentum method." J. Fluid Mech., 87(2), 335-341.
- Chwang, A. T., and Wang, K. H. (1984). "Nonlinear impulsive force on an accelerating container." J. Fluids Engrg., 106(2), 233-240.
- Faltinsen, O. M. (1974). "A nonlinear theory of sloshing in rectangular tanks." J. Ship Res., 18(4), 224-241.
- Kotsubo, S. (1959). "Dynamic water pressure on dams due to irregular earthquakes." Memoirs of the Fac. of Engrg., Kyushu Univ., Fukoka, Japan, 18, 119–129.
- Kotsubo, S. (1961). "External forces on arch dams during earthquakes." Mem. Fac. Eng., Kyushu Univ., Japan, 20, 327-366.
- Moiseev, N. N. (1958). "On the theory of nonlinear vibrations of a liquid of finite volume." J. Appl. Math. and Mech., 22, 860-872.
- Westergaard, H. M. (1933). "Water pressures during earthquakes." Trans., ASCE, 98, 418-433.

Computation of Viscous Flow around Series 60 Model and Comparison with Experiments*

Yusuke Tahara**(*Member*)

Ship boundary-layer and wake flows are analyzed by the finite-analytic scheme. The Reynolds-Averaged Navier-Stokes equations are solved with the standard $k - \varepsilon$ turbulence model and the wall-function approach. The numerical method developed by Patel et al.⁹ for calculating ship stern and wake flows has been modified and extended to analyze a wider flow region with a more general hull geometry. An overview is given for the present approach and numerical results are presented for the Series 60 model. The latter includes detailed comparisons between the present and previous¹⁰ computational results with available experimental data.

Keywords : Boundary-Layer Flow Navier-Stokes Equations $k - \varepsilon$ Turbulence Model Finite - Analytic Scheme, Series 60 Model

1. Introduction

Considerable effort has been put forth in the development of numerical methods to calculate ship boundary layer and wake flows. Many recent studies in this field have focused on the development of Navier-Stokes (NS) and Reynolds-Averaged Navier-Stokes (RANS) equation methods. Quite a few numerical schemes have been proposed ^{1) (2) 3)}. At present, the status of these methods is such that practical ship geometry can be considered, including complexities like appendages and propellers.

In the work of Patel et al.¹⁾, the numerical method developed by Chen and Patel⁴⁾ for the solution of the partially-parabolic RANS equations had been generalized to solve the fully-elliptic equations. Results presented for several ship forms indicated that many important features of the flow were adequately simulated. However, the solution domain is restricted to the ship stern and wake region, and proper inflow conditions must be specified at the inlet boundary.

Chen and Patel $^{5)}$ proposed some modifications

to the previous numerical method¹⁾, and applied them to calculate the flow around wing-body junctions. In this work, three important changes were made to the previous method¹⁾: (1) in place of the staggered grid, a regular(or collocated)grid was used ; (2) the two-step pressure-velocity coupling algorithm (SIMPLER) was replaced by a novel one- step procedure; and (3) in place of the wall functions, a generalized version of the two-layer approach to turbulence modeling of Chen and Patel⁶⁾ was employed.

Chen and Patel⁵⁾ found the following advantages based on the aforementioned modifications : (1) introduction of the regular grid reduced a number of grid geometry parameters that previously occupied the computer memory ; (2) the new pressure-velocity-coupling technique accelerated the solution convergence ; and (3) the generalized two-layer approach had a capability to capture the flow separation at a high Reynolds number. Also this method was used by Kim⁷⁾ and Kim⁸⁾ to investigate flow separation on a spheroid at incidence, and longitudinal vortices in turbulent boundary layers, respectively.

In the above-mentioned studies, it appears that the finite-analytic method used by Patel et al.¹⁾ and Chen and Patel⁵⁾ is a stable and useful numerical scheme to analyze a variety of flow fields. Fur-

^{*} Read at the Spring Meeting of Kansai Society of Naval Architects, Japan, May, 28, 1993, Received March 24, 1993

^{**} Iowa Institute of Hydraulic Research, The University of Iowa

thermore, the introduction of a regular grid and a new pressure-velocity-coupling method⁵⁾ is very attractive, because these modifications give the solution method more flexibility than the original method of Patel et al.¹⁾. However the two-layer approach used in the work of Chen and Patel⁵⁾ requires a large number of grids in the inner layer. If the flow separation is not expected, the wall-function approach used by Patel et al.¹⁾ might be preferable as far as the computational effort is concerned.

In this paper, a method proposed is a modified version of the method of Patel et al.¹⁾ by introducing a regular grid system and a new pressure-velocity-coupling technique developed by Chen and Patel⁵⁾. The RANS equations are solved with the standard $k - \varepsilon$ turbulence model and the two-point wall-function approach used by Patel et al.¹⁾ with some modifications based on the regular grid layout. Also, the solution domain is extended upstream of the bow in order to calculate the entire flow region and simplify the inflow conditions. In the following, an overview is given for the present numerical approach and results are presented for the Series 60 model, including comparisons between the present and previous¹⁾ calculations with available experimental data^{9) 10) 11)}.

2. Governing Equations

The non-dimensional RANS equations for unsteady, three-dimensional incompressible flow can be written in Cartesian tensor notation as

with

where $U_i = (U, V, W)$ and $u_i = (u, v, w)$ are the Cartesian components of mean and fluctuating velocities, respectively, normalized by the reference velocity U_0 , $x^i = (x, y, z)$ are the dimensionless coordinates normalized by a characteristic length L, p is the pressure normalized by ρU_0^2 , $Re = U_0 L/\nu$ is the Reynolds number, ν is the kinematic viscosity, and the barred quantities $-\overline{u_i u_j}$ are the Reynolds stress normalized by U_0^2 .

If the Reynolds stress $-u_i u_j$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity ν_i , i.e.

where $k = (\overline{uu} + \overline{vv} + \overline{ww})/2$ is the turbulent kinetic energy, equation (2) for the mean momentum becomes

where

Equations (1) and (4) can be solved for U_i and p when a suitable turbulence model is employed to calculate the eddy-viscosity distribution. In this study, the eddy viscosity distribution is given by

where ε is the rate of turbulent energy dissipation. k and ε are obtained from the transport equations

$$\frac{\partial k}{\partial t} + \sum_{j=1}^{3} \left(U_{j} - \frac{1}{\sigma_{k}} \frac{\partial \nu_{t}}{\partial x^{j}} \right) \frac{\partial k}{\partial x^{j}} - \frac{1}{R_{k}} \nabla^{2} k - G + \varepsilon = 0$$
....(7)
$$\frac{\partial \varepsilon}{\partial t} + \sum_{j=1}^{3} \left(U_{j} - \frac{1}{\sigma_{\varepsilon}} \frac{\partial \nu_{t}}{\partial x^{j}} \right) \frac{\partial \varepsilon}{\partial x^{j}} - \frac{1}{R_{\varepsilon}} \nabla^{2} \varepsilon - C_{\varepsilon 1} \frac{\varepsilon}{k} G$$

$$+ C_{\varepsilon 2} \frac{\varepsilon^{2}}{k} = 0$$
....(8)

when the effective Reynolds numbers R_k and R_{ε} are defined by

and

is the rate of production of k, and $(C_{\mu}, C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k, \sigma_{\varepsilon})$ are constants whose values are (0.09, 1.44, 1.92, 1.01, 1.3). It is convenient to rewrite the transport equations (4) through (10) for momentum (U, V, W) and turbulence quantities (k, ε) in the following general form:

$$\nabla^2 \phi = R_{\phi} \left[\sum_{j=1}^3 \left(U_j - \frac{1}{\sigma_{\phi}} \frac{\partial \nu_t}{\partial x^j} \right) \frac{\partial \phi}{\partial x^j} + \frac{\partial \phi}{\partial t} \right] + s_{\phi} \cdots (11)$$

where ϕ again represents any one of the convective transport quantities $(U, V, W, k, \varepsilon)$. The source functions s_{ϕ} for U_i , k and ε are, respectively,

 $s_k = -R_k(G-\varepsilon)$ (12b)

3. Discretization and Velocity-Pressure Coupling

We transform the physical solution domain into a rectangular region in the computational space (ξ^i, t) using the following coordinate transformations:

Then the continuity equation (1) and the transport equations (11) for momentum and the two turbulence parameters can be written as



$$\frac{1}{J}\sum_{i=1}^{3}\sum_{j=1}^{3}\frac{\partial}{\partial\xi^{i}}(b_{j}^{i}U_{j}) = 0 \qquad \cdots \cdots (14a)$$
$$\sum_{j=1}^{3}\left(g^{jj}\frac{\partial^{2}\phi}{\partial\xi^{j}\partial\xi^{j}} - 2a_{\phi}^{j}\frac{\partial\phi}{\partial\xi^{j}}\right) = R_{\phi}\frac{\partial\phi}{\partial\tau} + s_{\phi} \cdots (14b)$$

where

$$2a_{\phi}^{j} = \frac{R_{\phi}}{J} \sum_{n=1}^{3} b_{n}^{j} \left(U_{n} - \frac{\partial x^{n}}{\partial \tau} - \frac{1}{J\sigma_{\phi}} \sum_{m=1}^{3} b_{n}^{m} \frac{\partial v_{t}}{\partial \xi^{m}} \right) - f^{j}$$

$$\cdots \cdots \cdots (14c)$$

$$S_{\phi} = s_{\phi} - 2 \left(g^{12} \frac{\partial^{2} \phi}{\partial \xi^{1} \partial \xi^{2}} + g^{13} \frac{\partial^{2} \phi}{\partial \xi^{1} \partial \xi^{3}} + g^{23} \frac{\partial^{2} \phi}{\partial \xi^{2} \partial \xi^{3}} \right)$$

$$\cdots \cdots \cdots (14d)$$

The geometric coefficients b_i^j , g^{ij} , J and f^j appearing in the above equations are defined in Patel et al.¹⁾.

The five transport equations (14b) for (U_i, k, ε) are discretized by the finite-analytic scheme of Patel et al.¹⁾. In the finite-analytic scheme, equation (14b) is first linearlized in each local numerical element $(\Delta \xi^i, i=1, 2, 3)$ by evaluating the coefficients $2a_{\phi}^i$ and R_{ϕ} at an interior point P. The resulting linear equation is then solved analytically by the method of separation of variables. Evaluation of the analytic solutions at the interior node then provides a twelve-point discretization formula of the form of

$$\phi_{P} = \frac{1}{1 + C_{P} \left[C_{U} + C_{D} + \frac{R}{\Delta \tau} \right]} \left\{ C_{NE} \phi_{NE} + C_{NW} \phi_{NW} + C_{SE} \phi_{SE} + C_{SW} \phi_{SW} + C_{EC} \phi_{EC} + C_{WC} \phi_{WC} + C_{NC} \phi_{NC} + C_{SC} \phi_{SC} + C_{P} \left(C_{U} \phi_{U} + C_{D} \phi_{D} + \frac{R}{\Delta \tau} \phi_{P}^{n-1} \right) - C_{P} (S_{\phi})_{P} \right\}$$

$$(15)$$



Fig. 1 Nodes in regular grid and continuity cell

where $R = (R_{\phi})_P$ and the finite-analytic coefficients C_{nb} , C_P , etc. are given in Patel et al.¹⁾.

In the method of Chen and Patel⁵⁾ followed here, the pressure equation is derived by introducing pseudo-velocities at staggered locations while maintaining the regular grid arrangement for all the transport equations. Fig. 1 shows the locations of nodes in the regular grid in the ξ^2 - ξ^3 plane. All five transport quantities $(U, V, W, k, \varepsilon)$ and pressure are evaluated at the regular nodes (denoted by circles in the figure). In deriving the pressure equation, a control volume (shaded area in the figure) is employed as a continuity cell, to established the coupling between the velocity and pressure fields. The pressure equation used in this study is written as

$$(E_{d}^{11} + E_{u}^{11} + E_{n}^{22} + E_{s}^{22} + E_{e}^{33} + E_{w}^{33}) p_{\rho}$$

= $E_{d}^{11} p_{D} + E_{u}^{11} p_{U} + E_{n}^{22} p_{NC} + E_{s}^{22} p_{SC} + E_{e}^{33} p_{EC}$
+ $E_{w}^{33} p_{WC} - \widehat{D}$ (16a)

with

Here E^{ij} and a modified pseudovelocity \widehat{U}_i at the regular node are

$$E^{ij} = \frac{RC_P}{\int \left[1 + C_P \left(C_U + C_D + \frac{R}{\Delta \tau}\right)\right]} \sum_{m=1}^{3} b_m^i b_m^j \cdots (17)$$
$$\widehat{U}^i = \sum_{n=1}^{3} b_n^i \widehat{U}_n - E^{ij} \frac{\partial p}{\partial \varepsilon^j} - E^{ik} \frac{\partial p}{\partial \varepsilon^k} \qquad \cdots \cdots \cdots (18)$$

where \widehat{U}_i is a pseudovelocity given by the decomposition of equation (15) for U_i into \widehat{U}_i plus the pressure gradient terms, such that

$$U_{i} = \widehat{U}_{i} - \frac{RC_{P}}{J\left[1 + C_{P}\left(C_{U} + C_{D} + \frac{R}{\Delta\tau}\right)\right]} \stackrel{3}{\underset{j=1}{\overset{5}{\sum}} b_{i}^{j} \frac{\partial p}{\partial\xi^{j}} \cdots (19)$$

The coefficients E_d^{11} , E_u^{11} , etc. and the modified pseudovelocities \widehat{U}_d^1 , \widehat{U}_u^1 , etc. in equation (16) are defined at the staggered node, and obtained from those at the regular node by the one-dimensional linear interpolation.

The solution of the complete flow equations involves a global iteration process, in which the velocity-pressure coupling is effected by predictor-corrector steps. In the predictor step, the pressure field at the previous time step is used in the solution of the implicit equations (15) to obtain the corresponding velocity field. Since the velocity field generally does not satisfy mass conservation, a corrector step is needed. In the corrector step, the explicit momentum equations (19) and the implicit pressure equation (16) are solved iteratively to ensure the satisfaction of the continuity equation.



Fig. 2 Coordinate system

4. Boundary and Initial Conditions

Consider a ship fixed in the uniform onset flow $U_0 = (U_0, 0, 0)$ as depicted in Fig. 2. Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward. The solution domain is shown in Fig. 3. In terms of the notation of Fig. 3, the boundary conditions on each of the boundaries are described in the following.



Fig. 3 Solution domain and boundaries

On the inlet plane Si, the boundary conditions are provided on the basis of the freestream values, i.e.,

$$U = 1$$
, $V = W = \frac{\partial k}{\partial x} = \frac{\partial \varepsilon}{\partial x} = p = 0$ (20a)

On the exit plane Se, axial diffusion is negligible so that the exit conditions used are

On the symmetric plane Sc, the conditions imposed

are

In this study, free-surface effects are not considered, then the boundary conditions imposed on the waterplane Sw are similar to those on Sc, such that

On the body surface Sb, a two-point wall-function approach¹⁾ is used to give the boundary conditions for $(U, V, W, k, \varepsilon)$. Originally this approach was used with the staggered grid layout¹⁾, however much easier implementation is possible in the present regular grid layout. Zero normal gradient condition on Sb is used for p. On the outer boundary So, the conditions imposed are

$$U = 1$$
, $W = \frac{\partial k}{\partial r} = \frac{\partial \varepsilon}{\partial r} = p = 0$ (20e)

where r is the radial direction as shown in Fig. 3, and V is determined by the solution to satisfy the continuity equation.

In this study, the initial conditions are taken from the freestream values, i.e.,

$$U = 1, V = W = p = 0$$
(21)

For the turbulent quantities k and ε , very small values are initially specified. It is assumed that the flow is already turbulent upstream of the bow.

5. Overall Solution Procedure

The overall numerical solution procedure is summarized as follows:

- 1. Input the computational grid and calculate the geometric coefficients.
- 2. Specify the initial conditions for the velocity, pressure and turbulence fields.
- 3. Calculate the finite-analytic coefficients for the transport equation (15).
- 4. Solve equation (15) for the turbulence quantities (k, ε) .
- 5. Solve equation (15) for the velocities (U, V, W) using the previous pressure field (predictor stage for the velocity field).
- 6. Solve pressure equation (16).

- 7. Using the newly obtained pressure, calculate the new velocity field explicitly from equations (19) (corrector stage for the velocity field).
- 8. Repeat steps 6 and 7 for the specified number of times.
- 9. Return to step 3 for the next time step.

6. Body Geometry and Experimental Information

As already indicated, the computational results are presented for the Series 60 $C_B = .6$ ship model for which an extensive set of experimental data were recently obtained by Toda et al.⁹⁾. The experiments were performed at the Iowa Institute of Hydraulic Research (IIHR) towing tank for low and high Froude numbers, i.e., $Fr = U_0 / \sqrt{gL} = 0.16$ and 0.316, respectively. The principal dimensions and offsets of the model are given in Toda et al.⁹⁾. The model is 3.048-m between the perpendiculars, and constructed of the fiber-reinforced plexiglas. In order to simulate turbulent flow, a row of cylindrical studs of 1.6mm height and 3.2mm diameter were fitted with 9.5mm spacing on the model at x = 0.05. In the present study, the computational results are compared with the low Fr experimental data.

7. Results and Discussion

In the following, first the computational grids and conditions are described. Then some results are presented and discussed, including wherever possible comparisons with available experimental data. In the presentation of the results, variables are non-dimensionalized using the ship length L, the free-stream speed U_0 and fluid density ρ .

The computational grid was obtained using the technique of generating body fitted coordinates through the solution of elliptic partial differential equations, such that

Here ∇^2 is the Laplacian operator in Cartesian coordinates x^i as defined in equation (2b). In this study, the axial f^1 and circumferential f^3 control functions are specified as, respectively, only functions of ξ^1 and ξ^3 . However, in order to accurately satisfy the body-surface boundary condition and resolve the viscous flow, $f^2 = f^2(\xi^1, \xi^2, \xi^3)$,



(c) Body and wake crossplanes

Fig. 4 Computational grid

where ξ^1, ξ^2 , and ξ^3 are body fitted coordinates in axial, radial, and circumferential directions, respectively. A global view of the grid used in the computation is shown in Fig. 4a, while Figs. 4b and 4c show partial views of the grid for a longitudinal plane and typical body and wake crossplanes, respectively. The inlet, exit and outer boundaries are located at x = (-0.2, 3.9) and r=1, respectively. The first and second grid points off the body surface are located in the logarithmic law-of-the-wall region, i.e., $50 < y^+$ <500, where $y^+ = Re U_{\tau} y_n = Re \sqrt{\tau_w} / \rho U_0^2 y_n$ is the dimensionless distance measured in the direction normal to the surface, and τ_w is the wall-shear stress. The numbers of grid points used are 77, 30, and 15 in the axial, radial, and circumferential directions, respectively.

The conditions for the computation were as follows : L = 1 ; $U_0 = 1$; Re = 2,700,000. These conditions were selected to correspond as closely as possible to those of the experiments. The values of the



Fig. 5 Convergence history

time increment $\Delta \tau$ and underrelaxation factors for velocity $(\alpha_U, \alpha_V, \alpha_W)$, turbulence quantities $(\alpha_k, \alpha_{\epsilon})$ and pressure (α_p) were as follows : $\Delta \tau = 0.01$; $\alpha_U = \alpha_V = \alpha_W = 1$; $\alpha_k = \alpha_{\epsilon} = 0.5$; and $\alpha_p = 0.1$.

The computation was performed on a Cray Y-MP supercomputer. The CPU time required for the calculation was about 4 seconds for one global iteration. Fig. 5 provides the convergence history



Fig. 6 Pressure along the keel and waterline



Fig. 7 Pressure profile on the hull

for the pressure. The pressure residual is defined as

where i, it, and imax are the grid-point index, number of iteration, and total number of grid points, respectively. In the present study, the convergence criterion was that the change in solution be less than about 0.05 % for all variables, and it appeared that the present method satisfies this in 250 global iterations.

Figs. 6a and 6b show the variation of pressure along the keel and waterline, respectively. The experimental data of Toda et al.¹⁰⁾ are included in the figures. These measurements were made in a towing tank with the 4-m model at Fr = 0.16 and Re = 3,200,000. The data were obtained from the pitot-probe measurements on the port and starboard sides. The data on the waterline are those measured just below the free surface. It is seen that the agreement between the present results and the measurements is as good as that of Patel et $al.^{1)}$.

Fig. 7 shows the girthwise distribution of pressure at twelve stations. The girth length is measured from the waterline to keel, and non-dimensionalized using total girth length at each station. The experimental data from the University of Tokyo¹¹⁾ are included in the figure. Those measurements were made in a towing tank with the 4-m model at Fr = 0.18. It is clear that some wave effects are observed in the measurements, especially near the free surface. However, Toda et al.⁹⁾ noted from these data that wave effects are relatively small for such a low Fr, except near the bow and stern. Hence in the present study, the computed results are compared with the measurements in the
range 0.075 < x < 0.95, where x = 0 and 1.0 correspond to the FP and AP, respectively. Relatively good agreement between the computational and ex-

perimental results is observed.

Figs. 8 through 13 show the axial-velocity (U) contours, crossplane vectors (V-W), and pressure



Fig. 8 Comparison of solutions at x = 0: (a) Axial-velocity contours; (b) Crossplane-velocity vectors; and (c) Pressure contours

(*p*) contours for several representative sections, i.e., x = (0, 0.2, 0.4, 0.8, 1.0, 1.1), in which the experimental data of Toda et al.⁹⁾ at Fr = 0.16 are included for comparison. Figs. 14 through 19 provide detailed comparisons of the velocity and pressure profiles between the present computation



Fig. 9 Comparison of solutions at x = 0.2: (a) Axial-velocity contours; (b) Crossplane-velocity vectors; and (c) Pressure contours and the experiment⁹⁾ at the same stations. In Figs. 17 through 19, the computational results of Patel et $al_{\cdot}^{(1)}$ and the measurements of Toda et $al_{\cdot}^{(0)}$

(referred to as OSAKA in the figures) are also included. Those results are discussed in the following.



Experiment

Fig. 10 Comparison of solutions at x = 0.4: (a) Axial-velocity contours; (b) Crossplane-velocity vectors ; and (c) Pressure contours

Figs. 8 and 14 show comparisons of results at x=0. At this station, it is seen that the flow is dominated by stagnation effects associated with

the close proximity of the bow. The U contours of the computation are less than unity through the extent of the present region. The p contours of the



Fig. 11 Comparison of solutions at x = 0.8: (a) Axial-velocity contours; (b) Crossplane-velocity vectors ; and (c) Pressure contours

-66-

computation are similar to those for U, but with reverse trend in magnitude. The same trends of the U and p contours are observed in the measure-

ments. The computed V-W vectors show the double model stagnation flows. However, the measurements show downward shift of the stagnation



Fig. 12 Comparison of solutions at x = 1.0: (a) Axial-velocity contours; (b) Crossplane-velocity vectors; and (c) Pressure contours

-67-

point, which is due to bow-wave effects. The U profiles of the experiment display the appearance of laminar stagnation-point flow. The V profiles of measurements are positive at all depths due to outward displacement effects of the hull. The trends of the U and V profiles of the experiment are well



Fig. 13 Comparison of solutions at x = 1.1: (a) Axial-velocity contours;(b) Crossplane-velocity vectors ; and (c) Pressure contours

-68-

reproduced by the present computation. The measured W profiles are upward near the free surface and downward at larger depths, however the computed W profiles are downward at all depths due to the aforementioned reason. Both the computed and measured p profiles display quite large values in the inner part. At this station, overall agreement of results between the present computation and experiment is satisfactory.

Figs. 9 and 15 show comparisons of results at x = 0.2. At this station, U contours of the computation display the thickening of the boundary layer, especially near the keel, which is due to the flow convergence toward the keel. The measurements also show a similar trend, however, with more complexities. Both the computed and measured V-W vectors mostly show downward flow,

and an increased magnitude can be observed near the bilge. The measured V-W vectors show upward flow near the centerplane, however, this is not very clear in the computation. Such complex flow features are due to the bow-bilge vortex. The p contours of both the computation and experiment show the lowest values near the hull, especially in the region near the bow-bilge vortex. The computed U and V profiles show good agreement with the measurements. Both the computed and measured W profiles show similar trends, however, the computed W has a lower estimated magnitude near the bilge. Both the computed and measured p profiles are fairly uniform and show similar trends, although the computed p has somewhat higher estimated values at all depths.

Figs. 10 and 16 show comparisons of results at



Fig. 14 Velocity and pressure profiles at x = 0

x = 0.4. At this station, both the computed and measured U contours show the continued growth of the boundary layer and the convergence of the flow toward the keel, i.e., the boundary layer is quite thick near the keel, thin near the bilge, and somewhat thicker near the free surface. Both the computed and measured V-W vectors have decreased magnitude except near the bilge where the remnants of the bow-bilge vortex exist. The measured *p* contours display decreasing values from x = 0.2to the present section, except in the region near the bow-bilge vortex. This is also well reproduced by the present computation. At the present section, the computed U, V, W, and p profiles show very good agreement with the experimental data.

Figs. 11 and 17 show comparisons of results at

x = 0.8. At this station, the measured U contours display a pronounced bulge in the boundary laver near the region of maximum hull concavity and reduction in the boundary layer near the centerplane. The present computation shows a similar trend, however the above mentioned bulge in the boundary layer is not very clear. Both the computed and measured crossplane vectors are directed upward and toward the centerplane, in which the presence of a stern-bilge vortex is evident. Both the computed and measured p contours show negative values everywhere, with the lowest values in the region near the stern-bilge vortex. The measured Uprofiles display the detailed characteristics of the thick boundary layer in this section, which is also well reproduced by the present computation. For the V and W profiles, the present results show bet-



Fig. 15 Velocity and pressure profiles at x = 0.2

ter agreement with the experimental data than the results of Patel et al.¹⁾. The computed p profiles by both the present and previous¹⁾ methods show good agreement with the experimental data.

Figs. 12, 13, 18, and 19 show comparisons of results at x = 1.0 and 1.1. At these stations, the U contours of the present computation display merging of the boundary layer into the wake and its initial evolution, which is also observed in the measurements. However, the measurements show a relatively slower recovery near the free surface, which may be due to wave effects. For both the computation and experiment, the V-W vectors show that the flow is upward and toward the centerplane, in which the stern-bilge vortex evolves into a weak longitudinal vortex. The computed U profiles by the present and previous¹⁾ methods show faster recovery than the measurements at larger depths. It is clear that the present calculation did not improve one of the previous issues¹⁾, i.e., the difficulties in the $k - \varepsilon$ turbulence model with the wall-function approach. As is the case for the previous section, the computed V profiles by the present method show better agreement with the measurements than the previous method¹⁾, however, no particular improvement is observed in the W profiles. The computed p profiles by both the present and previous¹⁾ method show good agreement with the experimental data.

8. Conclusions

This work presents a numerical method for calculating ship boundary layer and wake flows. The computational results presented for the Series 60



Fig. 16 Velocity and pressure profiles at x = 0.4

ship model show satisfactory agreement with the experimental data. It appears that the present method is computationally more efficient than the method of Patel et al.¹⁾, and this validates one of conclusions of the work of Chen and Patel⁵⁾. Finally, some of the issues that must be addressed while further developing the present approach are as follows : improvement of accuracy in calculating wake flows ; introduction of a more appropriate turbulence model ; and further improvement of the computational grid. Also of interest is an extension of the present method for nonzero Fr, such as a work demonstrated by Tahara et al.¹².

9. Acknowledgments

The author wishes to thank Prof. Y. Toda at Kobe University of Mercantile Marine and Prof.

F. Stern, Prof. V. C. Patel, Mr. J. Longo, and many Postdoctoral Associates at the Iowa Institute of Hydraulic Research for their valuable discussions and encouragement. It is noted that the numerical work in this study has been carried out on a Cray Y-MP supercomputer at the San Diego Supercomputing Center, and again, the author expresses his thanks to the Weeg Computing Center of The University of Iowa for the allocation of supercomputing hours.

10. References

 Patel, V.C., Chen, H.C. and Ju, S., "Ship Stern and Wake Flows : Solutions of the Fully-Elliptic Reynolds-Averaged Navier-Stokes Equations and Comparisons with Experiments", Iowa Institute of Hydraulic Research,



Fig. 17 Velocity and pressure profiles at x = 0.8

The University of Iowa, IIHR Report No. 323, 1988 (unpublished) ; also J. of Computational Physics, Vol. 88, No. 2, June 1990, pp. 305-336.

- Proc. 1990 SSPA-CTH-IIHR Workshop on Ship Viscous Flow, Gothenburg, 1990.
- Proc. The Second Osaka International Colloquium on Viscous Fluid Dynamics in Ship and Ocean Technology, Osaka, 1991.
- Chen, H.C. and Patel, V.C., "Calculation of Trailing-Edge, Stern and Wake Flows by a Time-Marching Solution of the Partially-Parabolic Equations", Iowa Institute of Hydraulic Research, The University of Iowa, IIHR Report No. 285, 1985 (unpublished).
- 5) Chen, H.C. and Patel, V.C., "The Flow Around Wing-Body Junctions", Proc. 4th

Symp. Num. Phys. Aspects Aerodyn. Flows, Cebeci ed., Long Beach, CA, 1989.

- Chen, H.C. and Patel, V.C., "Near-Wall Turbulence Models for Complex Flows Including Separation", AIAA Journal, Vol. 26, 1988, pp. 641-648.
- Kim, S., "Numerical Studies of Three-Dimensional Flow Separation", Ph.D. Thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- Kim, W.J., "An Experimental and Computational Study of Longitudinal Vortices in Turbulent Boundary Layers", Ph.D. Thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- 9) Toda, Y., Stern, F. and Longo, J., "Mean-Flow Measurements in the Boundary



Fig. 18 Velocity and pressure profiles at x = 1.0

Layer and Wake and Wave Field of a Series 60 $C_B = 0.6$ Ship Model for Froude Numbers .16 and .316", IIHR Report No. 352, 1991 (unpublished) ; also J. of Ship Research, Vol. 36, No. 4, 1992, pp. 360-377.

- 10) Toda, Y., Stern, F., Tanaka, I., and Patel, V. C., "Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B = 0.6 Model Ship With and Without Propeller", IIHR Report No. 326, 1988 (unpublished); also J. of Ship Research, Vol. 34, No. 4, 1990, pp. 225-252.
- ITTC, "Report of the Resistance and Flow Committee", Proc. 18th International Towing Tank Conference, Kobe, Japan, 1987, pp. 47-92.
- 12) Tahara, Y., Stern, F. and Rosen, B., "An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number", Proc. 18th Symposium on Naval Hydrodynamics, Ann Arbor, 1990, pp. 699-720; also J. of Computational Physics, Vol. 98, No. 1, January 1992, pp. 33-53.



Fig. 19 Velocity and pressure profiles at x = 1.1



0045-7930(94)E0002-F

Computers Fluids Vol. 23, No. 6, pp. 785-816, 1994 Copyright © 1994 Elsevier Science Ltd Printed in Great Britain. All rights reserved 0045-7930/94 \$7.00 + 0.00

VALIDATION OF AN INTERACTIVE APPROACH FOR CALCULATING SHIP BOUNDARY LAYERS AND WAKES FOR NONZERO FROUDE NUMBER

Y. TAHARA and F. STERN

Department of Mechanical Engineering, Institute of Hydraulic Research, The University of Iowa, Iowa City, IA 52242-1585, U.S.A.

(Received 11 June 1993; in revised form 17 November 1993)

Abstract—An interactive approach for calculating ship boundary layers and wakes for nonzero Froude number is validated through detailed comparisons with recent extensive experimental data for the practical 3-D geometry of the Series 60 $C_{\rm B} = 0.6$ ship model at both low (0.16) and high (0.316) Froude number. The former case essentially simulates the zero Froude number condition such that the comparisons with the latter case enables the identification of the salient features of the wave-induced effects. Close agreement is demonstrated between the calculations and the data, which supports the conclusion that the present approach can accurately predict ship boundary layers and wakes, including free-surface effects; however, the present detailed comparisons enables a more critical evaluation. Additionally, comparisons are made with inviscid-flow results, which in combination enable an evaluation of the wave/boundary-layer and wake interaction. Lastly, some concluding remarks are made concerning the limitations of the present approach, prognosis and requirements for improvements, relative merits of interactive vs large-domain approaches, and implications with regard to ship design.

NOMENCLATURE

 $C_{\rm B} = {\rm block}$ coefficient

 $C_{\rm p} = {\rm pressure \ coefficient} \ (= 2\hat{p}/\rho U_o^2)$

- $C_{\rm R}$ = residuary-resistance coefficient (= $2R/\rho SU_{\rm e}^2$)
- d = draft
- Fr = Froude number $(=U_o/\sqrt{gL})$ H = total head $[=(C_p + u^2 + v^2 + w^2)^{1/2}]$
- k = turbulent kinetic energy
- L = characteristic (ship) length
- p = static pressure
- \hat{p} = piezometric pressure
- R = residuary resistance
- Re = Reynolds number $(=U_a L/v)$
- S = wetted surface area
- $S_{\rm b}, S_{\rm e},$ etc. = boundaries of the solution domain
- u, v, w = velocity components in cylindrical polar coordinates
 - $U_{\rm c}$ = wake centerline velocity
 - $U_o =$ characteristic (freestream) velocity
 - U_{τ} = wall-shear velocity (= $\sqrt{\tau_{w}/\rho}$)
 - v-w = crossplane vectors
 - $c, r, \theta = cylindrical polar coordinates$
 - X, Y, Z =Cartesian coordinates
 - v^+ = dimensionless distance (= $U_\tau y/v$)
 - $\alpha_{t}, \alpha_{p},$ etc. = under relaxation factors
 - δ = boundary-layer and wake thickness
 - $\delta^* = displacement thickness$
 - ϵ = rate of turbulent energy dissipation
 - v = kinematic viscosity
 - ζ = free-surface elevation
 - $\xi, \eta, \zeta =$ body-fitted coordinates
 - $\rho = \text{density}$
 - $\omega_x = axial \text{ vorticity}$
 - $\tau_{\rm w}$ = wall-shear stress
 - $\hat{\phi}$ = transport quantities (u, v, w, k, ϵ)

-75-

INTRODUCTION

Although considerable effort has been put forth in the development of viscous-flow methods for calculating ship boundary layers and wakes such that practical geometries can be considered, including appendages and propellers [e.g., 1–4], relatively little attention has been given to the inclusion of free-surface effects, i.e., most methods neglect such effects and solve the so-called zero Froude number (Fr) problem in which the free-surface is a symmetry plane. The studies that have been conducted are mostly of an approximate nature with only a few exceptions, i.e., the large-domain approaches of most recently [5-8] and the interactive approaches of [9] and very recently [10, 11]. See [9] for a more complete review, including additional references.

This paper is concerned with validation of the interactive approach of [9] through detailed comparisons with recent extensive experimental data for the practical 3-D geometry of the Series $60 C_B = 0.6$ ship model at both low (0.16) and high (0.316) Fr [12, 13]. The former case essentially simulates the zero Fr condition such that the comparisons with the latter case enables the identification of the salient features of the wave-induced effects. Additionally, comparisons are made with inviscid-flow results [13], which in combination enable an evaluation of the wave/boundary-layer and wake interaction. Lastly, some concluding remarks are made concerning the limitations of the present approach, prognosis and requirements for improvements, relative merits of interactive vs large-domain approaches, and implications with regard to ship design. From the outset it should be recognized that the large-domain approach is also of interest and currently under development the status of which will also be discussed in the Concluding remarks. In [9], results and similar comparisons were presented for the Wigley hull; however, the limited available data precluded a complete assessment.

INTERACTIVE APPROACH

The approach modifies and extends two of the leading inviscid- [14] and viscous-flow [15] methods for interactive calculations for ship boundary layers and wakes for nonzero Fr: the Reynolds-averaged Navier–Stokes (RaNS) equations are solved using a small domain with edge conditions matched with those from a source-doublet-Dawson method solved using a displacement-body. The details of both methods, including the necessary modifications and extensions, are provided in [9]. The viscous-flow solution domain and inviscid-flow displacement body are shown in Figs 1 and 2, respectively.

The viscous-flow method solves the unsteady 3-D RaNS equations in conjunction with the continuity equation and the $k-\epsilon$ turbulence model for the mean-velocity components (u, v, w), pressure p, and turbulent kinetic energy k and its dissipation-rate ϵ . The transport equations for $\phi = (u, v, w, k, \epsilon)$ are written in cylindrical-polar coordinates (x, r, θ) in the physical domain, and partially transformed (i.e., the coordinates only and not the velocity components) into numerically generated, body-fitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) such that the computational domain forms a simple rectangular region with equal grid spacing. A SIMPLER-type velocity-pressure coupling algorithm is used. The transformed equations are discretized using a staggered grid and the finite-analytic method, and solved using the method of lines. The equations are solved in unsteady form; however, for steady-flow applications, time t simply serves as an iteration parameter.

Referring to Fig. 1, the specified boundaries of the solution domain are the body surface S_b , the inlet plane S_i , the exit plane S_e , the symmetry plane S_k , the outer boundary S_o , and the free-surface S_{ζ} . The boundary conditions are as follows: on S_b , a two-point wall-function is used; on S_i , the inlet conditions are specified from the inviscid-flow solution for (u, v, w, p) and from typical free-stream values for (k, ϵ) ; on S_e , axial diffusion and pressure gradient are assumed negligible, i.e., $\phi_{\xi\xi} = p_{\xi} = 0$; on S_k , the conditions imposed are $(u, k, \epsilon, p)\zeta = w = 0$; on S_o , the edge conditions are specified from the inviscid-flow solution for (u, w, p), v is obtained from the continuity equation, and $(k, \epsilon)_{\eta} = 0$; and on S_{ζ} , inviscid first-order (in ζ) approximations to the free-surface boundary conditions are specified on the fixed surface S_w , i.e., the dynamic conditions reduce to $p = \zeta/Fr^2$ and $(u, v, w, k, \epsilon)_{\zeta} = 0$, and the kinematic condition is used to obtain ζ with inlet, exit, and outer boundary values specified from the inviscid-flow solution [see equations (18)–(20) of [9]}.

The inviscid flow is calculated using SPLASH: an extended version of the source-doublet panel method of Maskew [16, 17], originally developed for the prediction of subsonic aerodynamic flows, about arbitrary configurations, modified to include the presence of a free surface and gravity waves both for submerged and surface-piercing bodies, through the addition of fixed free-surface panels on S_w and first-order free-surface boundary conditions. As is the case with [16, 17], lifting surfaces and their associated wake treatments as well as wall boundaries are included; however, the present calculations are for nonlifting unbounded flow.

The interaction procedures are based on extensions of those developed previously for zero Fr [18], i.e., the interaction law is based on the concept of displacement thickness δ^* , and the match boundary (i.e., S_0) is located at about 2δ , where δ is the boundary-layer and wake thickness. An



Fig. 1. Computational grid and definition sketch of flow-field regions and solution domains. (a) Longitudinal plane and (b) body crossplane.

-77-



-78-

approximate definition is used for δ^* in which 2-D streamwise thicknesses at each axial station along η grid lines are connected by a cubic spline.

In the figures and discussions, the coordinates, velocities, pressure, etc. are nondimensionalized using L, with x = 0 at the body leading edge, U_0 , and fluid density ρ . Also, the terminology "interactive" refers to results from both the interactive viscous and displacement-body inviscid solutions. When the distinction is not obvious it will be made. The terminology "inviscid" or "bare-body" refers to the noninteractive inviscid solution.

Lastly, it should be recognized that certain advancements have been made in both the viscous-(for other applications) and inviscid-flow methodologies concurrently with the present work; however, these were either unavailable or not deemed necessary to reach the present conclusions. For example, with regard to the former, regular grids with PISO- or MAC-type velocity-pressure coupling, high-order finite-difference schemes for space discretization, near-wall turbulence modeling, time-accurate unsteady-flow computations, and CHIMERA overlaid domain decomposition grid generation [19, 20, 21], and with regard to the latter, nonlinear free-surface boundary conditions [22]. Issues concerning some of these advancements and their influences are currently being addressed and will be discussed in the Concluding remarks.

SERIES 60 $C_{\rm B}$ = 0.6 SHIP MODEL EXPERIMENTAL DATA

Recently, extensive experimental data was obtained for the Series 60 $C_{\rm B} = 0.6$ ship model at the Iowa Institute of Hydraulic Research towing tank in order to explicate the influence of wavemaking by a surface-piercing body on its boundary layer and wake and provide detailed documentation of the complete flow field appropriate for validating computational methods [12, 13]. The series 60 $C_{\rm B} = 0.6$ ship model was selected for the experiments as a representative fine hull form and to complement the many previous and ongoing studies with this geometry. In particular, is the fact that it is one of the four hull forms selected for the Cooperative Experimental Program of the International Towing Tank Conference [23].

Mean-velocity and pressure field measurements were performed for Fr = 0.16 (average Reynolds number $Re = U_0 L/\nu = 2.22 \times 10^6$) and 0.316 (average $Re = 4.34 \times 10^6$) for a 3.048 m Series 60 $C_B = 0.6$ hull form at numerous stations from the bow to the stern and into the near wake. Additionally, wave profiles and local and global elevations were measured and resistance tests were conducted, including results for a 1.929 m model (for Fr = 0.316, average $Re = 2.44 \times 10^6$) for an evaluation of scale effects on near-field wave patterns. Lastly, comparisons were made between the data and SPLASH calculations in order to evaluate the capabilities of inviscid theory.

Unfortunately, this program of experiments did not include surface-pressure measurements; however, surface-pressure and wave profile measurements were made at similar Fr in [23], i.e., at the University of Tokyo for Fr = 0.18 and 0.3. In [12], it was concluded that the gross features of the flow fields for Fr = 0.18 and 0.3 are similar to those for Fr = 0.16 and 0.316, respectively, based on a comparison of the respective wave profiles.

Detailed information concerning the experiments, including uncertainty analysis and discussion of the results, is provided in [12, 13]. Herein, comparisons are made between the present calculations and the data and inviscid theory wherever possible.

COMPUTATIONAL GRID AND CONDITIONS

A partial view of the grid used for the viscous-flow calculations is shown in Fig. 1. Initially, a large-domain grid was generated, from which a small-domain grid was obtained by simply deleting that portion of the large-domain grid that lay beyond about r > 0.2. The inlet, exit, and outer boundaries are located at x = (-0.01, 4.88) and r = 1 (large domain) or r = 0.2 (small domain), respectively. The first grid point off the body surface is located in the range $90 < y^+ < 250.68$ axial, 30 (large domain) or 21 (small domain) radial, and 19 circumferential grid points were used. In summary, the total number of grid points for the large- and small-domain calculations are 38,760 and 27,132, respectively.

The inviscid-flow displacement-body and free-surface panelization are shown in Fig. 2. 423 panels are distributed over the displacement body and 546 over the free surface for a total number of 969 panels. The panelization covers an area corresponding to 1 ship length upstream of the bow,



790

-80-



Fig. 4. Wave profiles.

1.5 ship lengths in the transverse direction, and 3 ship lengths downstream of the stern. This panel arrangement was judged optimum based on panelization dependency tests [13].

Steady-flow calculations were performed for conditions corresponding to the experiments, i.e., Fr = 0 and 0.316 and $Re = 3.28 \times 10^6$, the latter being the average value for both Fr. Initially, both large-domain and interactive small-domain calculations were performed for zero Fr. In both cases, the calculations were started with a zero-pressure initial condition and freestream edge and inlet conditions and the values for the time α_t , pressure α_p , and turbulent kinetic energy and dissipation rate $\alpha_{k \text{ or } c}$ underrelaxation factors are as follows: $\alpha_t = 0.3$; $\alpha_p = 0.2$; and $\alpha_{k \text{ or } c} = 0.2$. In the case of the small-domain calculations, after 200 global iterations, the edge and inlet conditions were updated from the inviscid-flow solution using the latest values of displacement thickness. Subsequently, the edge and inlet conditions and displacement thickness were updated every 200 global iterations until convergence was achieved, which took three updates.

Then small-domain calculations for nonzero Fr were started with the zero Fr solution as the initial condition, and with nonzero Fr edge conditions obtained utilizing the zero Fr displacement body. This solution converged in 250 global iterations, and it appeared that the updated nonzero Fr displacement body was almost identical to that for zero Fr, such that the differences between new and previous edge conditions were very small. Therefore, no further interactions were carried out for nonzero Fr. The values for α_t , α_p and $\alpha_{k \text{ or } c}$ are the same as those for the small domain calculations for zero Fr.

The calculations were performed on the Naval Oceanographic Office and National Aerodynamic Simulation Program CRAY supercomputers. The average job run CRAY hours and central memory were 0.58 hours and 1.5 mw for 200 global iterations for the viscous-flow code and 0.016 hours and 0.75 mw for the inviscid-flow code. Extensive grid dependency and convergence tests were not carried out for the present application; since, these were done previously both for zero Fr [15] and other applications. Based on these tests along with the overall results of [1] and SUBOFF in which, in some cases, much finer grids were used, the grid was considered sufficiently refined. The convergence criterion was that the residual {see equation (36) of [9]} for all variables be about 10^{-4} . The convergence characteristics were similar to those shown previously for the Wigley hull (see Fig. 4 of [9]).

RESULTS

In the following, results are presented and discussed for Fr = 0 and 0.316, including comparisons with the data and, in the latter case, with inviscid theory. Figure 3 provides a comparison of certain aspects of the interactive and large-domain solutions for Fr = 0 and the interactive solution for Fr = 0.316. The wave profiles (cf. Fig. 4 of [12]), wave-elevation and wave-slope contours (cf. Figs 5 and 6 of [12]), free-surface perspective view, and transverse and longitudinal wave-elevation profiles (cf. Figs 6 and 7 of [13]) are shown in Figs 4–8, respectively. Surface-pressure profiles and surface-pressure and axial and vertical gradient contours are shown in Figs 9–12, respectively (cf. Figs 9–12 of [12]). Lastly, total-head, axial-velocity, pressure, axial-vorticity, and turbulent kinetic energy contours and crossplane vectors (cf. Fig. 7 of [12]) and total-head, velocity, and pressure



Fig. 5. Wave-elevation and -slope contours.

-82-

profiles (cf. Fig. 8 of [12]) are shown in Figs 13 and 14, respectively. The corresponding inviscid-theory results are provided in Figs 8–12 of [12] and 2 and 6–11 of [13]. The discussions are based on the complete results at all experimental locations (see Fig. 3 of [12]), although only representative results are shown. For brevity of presentation, in most cases, the data and inviscid theory are not reproduced, but rather, a similar format is used in the presentations as in [12, 13] to facilitate the comparisons with the corresponding already referenced figures. In general, the emphasis of the discussions concerns the comparisons and evaluation of the computational method, whereas reference is made where appropriate to [12, 13] for relevant discussion and interpretation of the flow physics; however, in some cases, for clarity of presentation, certain discussions are replicated.

Fr = 0

Figure 3 provides a comparison of the zero Fr large-domain and interactive solutions. The two solutions are nearly identical, which further validates the present interactive approach, beyond that already established for simple geometries and afterbody flow in [9] and [18], for the present practical ship geometry, including both the forebody and afterbody flow. Some additional aspects of the zero Fr interactive solution are displayed in Figs 9-14 for comparison with the data and later with the nonzero Fr solution. In summary, the flow exhibits the well known features for zero Fr for this type of hull form. At the bow, the flow is dominated by stagnation effects and, as is also the case on the forebody, by the displacement effects associated with the increasing cross sections of the hull and associated divergence of the inviscid streamlines (i.e., the inviscid-flow displays favorable axial surface-pressure gradients and downward crossplane vectors and insignificant viscous-inviscid interaction), which leads to the development of a thin boundary layer, except near the keel where there is a thickening due to the flow convergence, and the formation of a bow-bilge vortex. On the afterbody, the flow is dominated by the effects associated with the decreasing cross sections of the hull and the associated convergence of the inviscid flow (i.e., the inviscid-flow displays adverse axial surface-pressure gradients and upward crossplane vectors and significant viscous-inviscid interaction), which leads to a rapid thickening of the boundary layer with a characteristic bulge in the midgirth region, except near the keel where there is a thinning due to the flow divergence off the centerplane, and the formation of a stern-bilge vortex. In the near and intermediate wake, the inviscid-flow recovers the uniform stream and the viscous-inviscid interaction diminishes with axial distance, which leads ultimately to the development of an axisymmetric wake.

The fact that the large-domain afterbody flow solution generally indicates close agreement with the data was already established in [15] for similar grids and conditions. It was concluded that all the essential features of the mean flow could be predicted with considerable accuracy, including the pressure and wall-shear stress distributions on the hull, the boundary-layer thickness, and the mean-velocity field at the stern; however, it was also pointed out that certain details of the flow in the extreme stern and near-wake regions were not captured. Subsequently, in [4], the large-domain afterbody flow solution, including a modified $k-\epsilon$ turbulence model, was more generally and critically examined, as an aid in quantifying the current limitations of computational fluid dynamics (cfd) for predicting propeller-hull interaction. It was concluded that the modified $k-\epsilon$ turbulence model leads to an accurate prediction of the 0th harmonic and some improvement was evident for certain higher harmonics and other details of the nominal propeller inflow; however, clearly there was a limitation in the ability to predict the higher harmonics and very detailed flow. These level of differences were considered a general assessment with regard to the current status of cfd; since, the present computational method has been shown to be one of the leading current methods (e.g., SSPA-CTH-IIHR [1] and SUBOFF Workshops). Although some improvements could be expected through the use of, e.g., the earlier mentioned advancements in viscous-flow methodologies, clearly significant advancements in all these areas are required for very detailed resolution of the flow.

The present Fr = 0 large-domain and interactive solutions support these conclusions, including the forebody flow. An evaluation of cfd for nonzero Fr calculations will be given in the Concluding remarks.



794

Y. TAHARA and F. STERN

- 84 --





Fig. 8. Wave-elevation profiles: longitudinal.

- 86 -





Fr = 0.316

Figure 3 also includes nonzero Fr results for comparison. On the waterline, the surface and wake centerplane pressure displays large differences. The trends for the wall-shear velocity are similar, but with reduced magnitude. The wake centerline velocity indicates slower recovery near the stern, and faster recovery in the intermediate wake. These trends correlate with the wave profile (cf. Fig. 4), boundary-layer growth (cf. Fig. 13), and wave-induced pressure gradients (cf. Fig. 5), respectively. On the keelline, only relatively small differences are observed.

The wave profile at the hull is compared with the data in Fig. 4. The results demonstrate fairly close agreement with the data with respect to wave amplitude, shape, and phase, although some systematic differences are observed: the fullness of the bow wave is underpredicted; the location

CAF 23/6---D



Fig. 10. Surface-pressure contours.

of the shoulder-wave troughs and crest between them is overpredicted by about 8% L; and the stern-wave amplitude is somewhat underpredicted.

The wave-elevation and -slope contours are shown in Fig. 5. The elevations show close similarity with the data with regard to both amplitude and shape for the forebody, except at the bow where the bow-wave amplitudes are underpredicted. The results for the afterbody also show fairly close similarity; however, the detailed resolution particularly of the complex global-region wave system is incomplete and the stern-wave crest amplitudes are somewhat underpredicted. The slopes, which are indicative of the axial and transverse wave-induced pressure gradients, respectively, in the vicinity of the free surface, are also similar to the data in displaying a similar pattern to the elevations, except for the expected phase shift, but underscore the lack of detailed resolution. Figure 6 provides a free-surface perspective view. Note that the interactive results shown in Figs 5, 6, and 7 (the latter will be discussed next) are for the combined inviscid and viscous calculations and thus clearly display the smooth matching of the two solutions.

The transverse and longitudinal wave-elevation profiles are compared with the data in Figs 7 and 8, respectively. The calculations were interpolated from the computational free-surface grid (Figs 1 and 2) onto the measurement x and y stations using 3-D splines. First, the transverse profiles



Fig. 11. Axial surface-pressure gradient contours.

are considered. The results on the forebody show good agreement with both the global and local data, although, in the latter case, the bow-wave crest and the shoulder-wave trough are significantly and somewhat underpredicted, respectively. On the afterbody, the calculations also show fairly good agreement with the data; however, the amplitudes of the crests and troughs in the global region and the stern-wave crest are somewhat underpredicted, respectively. In many respects the calculations show remarkable similarity to the data, but clearly lack the complexity of the stern-and global-region wave systems displayed by the data and have the tendency of smoothing the indicated trends, i.e., in general, the wave slopes are not accurately simulated. The results for the longitudinal profiles are consistent with the previous discussions: the results underpredict the shoulder-wave troughs and crest between them; somewhat underpredict the stern-wave amplitudes; and underpredict the amplitudes of the crests and troughs in the global region. The differences increase with distance from the hull, which may be attributed to the panelization in this region and the wave-damping mechanism near the outer boundary employed in the inviscid-flow method.

Next, we consider the comparisons for the surface-pressure distribution. The surface-pressure profiles and the surface-pressure and axial and vertical surface-pressure gradient contours are



Fig. 12. Vertical surface-pressure gradient contours.

shown in Figs 9 and 10, respectively. The profiles show close similarity to the data, especially in consideration of the differences in Fr, which, unfortunately, precludes a quantitative comparison, i.e., the corresponding data is for Fr = 0.3 and 0.18, respectively (see [12] for relevant discussion concerning Fr differences). Most of the trends described in [12] with regard to the data are evident in the calculations. However, the Fr = 0.316 calculations appear to underpredict and overpredict the pressure rise at the bow near the free surface and keel, respectively. The location of the midbody pressure oscillations is shifted aft. Note that these differences are consistent with those described earlier for the wave profile. The contours also show fairly close similarity, including similar regions of favorable and adverse pressure gradients, with the exception of the aforementioned differences. Note that the values for the calculated pressure–resistance coefficients for the interactive and inviscid solutions were 1.807 and 1.526×10^{-3} , respectively, whereas the experimental residuary-resistance coefficient C_R (i.e., total–frictional resistance) value was 2.462×10^{-3} . In the former case, a part of the difference may be attributed to the Re differences, i.e., 3.28×10^6 vs 2.44×10^6 between the calculations and experiment, respectively.

Finally, with regard to the comparisons of the calculations and the data, we consider the results for the mean-velocity and pressure fields. The total-head, axial-velocity, pressure, axial-vorticity,

800



-91-



802

-92-

٠



— 93 —



-94-

804

Interactive approach for calculating ship boundary layers and wakes



-95-

806 Y. TAHARA and F. STERN 1.00 1.00 80 0 0 -0.0100 Z -0.0100 Z= Η D 1.00 1.00 10000 0 Z=-0.0200 Z=-0.0200 6<u>6908</u>-8 1.00 1.00 80\00 0 Z=-0.0300 Z=-0.0300 **200** 1.00 8 Z=-0.0400 Z=-0.0400 1.00 1.00 100 0 0 Z=-0.0500 Z=-0.0500 0.50 0.50 0.00 0.04 0.08 0.12 0.00 0.04 0.08 0.12 0.00 0.00 000 Z=-0.0100 Z=-0.0100 \geq Μ 0.00 0.00 -0.0200 Z =Z=-0.0200 0.00 0.00 <u>. R Sy S</u> Z=-0.0300 Z=-0.0300 0.00 0.00 Z=-0.0400 Z=-0.0400 0.00 0.00 Z=-0.0500 Z=-0.0500 888-8-8 -0.20 -0.20 0.00 0.04 0.08 0.12 0.00 0.04 0.08 0.12 Y 0.20 0.00 CD CD Z=-0.0100 Calculation (Fr = 0.316) Z=-0.0200 Calculation (Fr = 0)0 Experiment IIHR (Fr = 0.316) 0.00 Δ Experiment IIHR (Fr = 0.160) Z=-0.0300 0.00 Z=-0.0400 000 0.00 Z=-0.0500 -0.20 0.00 0.04 0.08 0.12 Y

Fig. 14 (a)—caption on p. 810.



Fig. 14 (b)-caption on p. 810.

0.12

Z=-0.0500

0.08

Y

-0.20

0.00

0.04

807



Fig. 14 (c)-caption on p. 810.


Fig. 14 (d)-caption overleaf.





and turbulent kinetic energy contours and crossplane vectors and total-head, velocity, and pressure profiles are shown in Figs 13 and 14, respectively. The order of discussion follows that used for the data in [12].

At x = 0, the general features of the stagnation effects are accurately predicted, i.e., the shapes of the u and p contours and crossplane flow, including location of the stagnation point are similar to the data (i.e., the u values are significantly larger near the free surface such that the contours are curved back towards the centerplane, which results from the reduced magnitude of $2\zeta g/U_0^2$ at the bow and associated reduction in adverse axial-pressure gradient upstream of the bow; downward shift of the stagnation point and decreased and increased crossplane flow below and above the stagnation point, respectively; and the p contours are similar to those for u, but with reverse trend in magnitude), however, the region of low velocity and high pressure and magnitude of the crossflow is underpredicted. The magnitude of ω_x is small with positive and negative values above and below the stagnation point, respectively. The profiles indicate that the agreement with the trends for the data, at least for the outer region, is very good (i.e., the u profiles have larger values than for Fr = 0.16, but the differences reduce with depth; the v profiles are similar for both Fr, which is consistent with the small ζ_y variations at this location; the w values are larger for all depths, which correlates with ζ_x ; and the trends for p are similar to those for u, but with reverse trend in magnitude). This is true for all three velocity components and the pressure, and at all depths. However, systematic differences are observed for the inner region, i.e., u, v, and w are underpredicted and p is overpredicted. These differences reduce with depth. The fact that the u and w differences can be correlated (i.e., display in-phase relationships) with ζ and ζ_x , respectively, and that the v differences are related to ζ_{v} will also be pointed out at the other stations and discussed further later in the evaluation of the wave/boundary-layer and wake interaction.

At x = 0.1, 0.2, and 0.4, the H contours indicate that δ is similar for both Fr, which was also the case for the data. At x = 0.1 and 0.2, the u contours display a broad region where u < 1 due to combined displacement effects and pronounced effects of the bow wave, but with lower and higher values than the data in the inner and outer regions, respectively. At x = 0.4, the u contours indicate increased values in the outer region due to the effects of the forebody shoulder-wave trough, but with reduced values in comparison to the data. The trends for v-w are similar to those for the data (i.e., at x = 0.1, larger downward flow near the hull and upward flow in outer region; at x = 0.2 and 0.4, larger downward flow, especially near the free surface; and the formation of a bow-bilge vortex), although the flow very close to the hull surface is unresolved, including the bow-bilge vortex. Both the trends and values for the p contours are similar to the data (i.e., at x = 0.1 and 0.2, broad region of high p due to bow wave; and at x = 0.4, lower values and contours parallel to free surface due to forebody shoulder-wave trough). $\omega_x < 0$ and similar for both Fr. The k contours correlate with those for H. The H profiles are similar to the data in indicating that δ is reduced for Fr = 0.316, but indicate a thicker boundary layer than the data for both Fr = 0 and 0.316. At x = 0.1 and 0.2, the *u* profiles indicate smaller values for Fr = 0.316, which correlates with ζ , as did the data, but, here again, indicate that δ is larger for the calculations than for the data. At x = 0.4, the trends and values for the u profiles are similar to those for the data (i.e., increased values, which correlate with ζ and are due to the effects of the forebody shoulder-wave trough), including a better agreement for δ . The v and w profiles also show similar trends as the data (i.e., the v values are reduced and increased, respectively, in the inner and outer regions at x = 0.1 and 0.2 and show minimal differences at x = 0.4, which correlates with ζ_{y} ; and the w values are reduced and increased in the inner and outer regions, respectively, at x = 0.1 and reduced overall at x = 0.2 and 0.4, which correlates with ζ_x), but, as was the case with v-w, the flow close to the hull surface is unresolved, including the bow-bilge vortex, and the downward flow at x = 0.4is overpredicted. Consistent with the p contours, the p profiles display close agreement with the data (i.e., increased values at x = 0.1 and 0.2; and at x = 0.4, reduced values). The Fr differences for the velocity and pressure profiles reduce with depth, as was also the case with the data.

At x = 0.6, 0.8, and 0.9, the *H* contours display minimal Fr differences, whereas the data indicated a reduction in δ and an increase in the mid-girth bulge at the latter two stations. At x = 0.6, the *u* contours also display minimal Fr differences, in contrast to the data, which displayed reduced values in the outer region and the reduction in δ . At the latter two stations, the *u* values are larger for Fr = 0.316, but the differences are not as large as those for the data and, as was the

case for Fr = 0, the mid-girth bulge is not predicted and, therefore, nor is its increase for Fr = 0.316. The trends for v-w are similar to the data (i.e., at x = 0.6, increased upward flow; at x = 0.8 and 0.9, reduced upward flow and increased convergence towards the centerplane; and formation of the stern-bilge vortex), except that the stern-bilge vortex is not well defined. The p contours indicate smaller and larger values at x = 0.6 and 0.8, respectively, in reverse trend to the data, and similar values at x = 0.9. The differences at the former two stations correlate with the differences between the measured and predicted afterbody shoulder-wave trough. $\omega_{\rm r}$ is similar to the data (i.e., minimal Fr effects) and, here again, k correlates with H. The H and u profiles display similar trends as the data (i.e., reduction in δ near the free surface and keel; at x = 0.6, u is reduced in the outer region; at x = 0.8 and 0.9, u is increased; and all of which correlate with ζ), but with less differences and increased δ . The trends for the v and w profiles are also similar to those for the data (i.e., at x = 0.6, the v values are similar, whereas at x = 0.8 and 0.9, they are reduced, all of which correlate with ζ_{v} ; and at x = 0.6 and 0.9, the w values are increased, whereas at x = 0.8, they are increased, all of which correlate with ζ_x); however, at x = 0.6 and 0.9, the profiles display lower magnitudes than the data, whereas at x = 0.9, the magnitudes are larger. In general, the inner flow is not resolved. As already noted with regard to the contours, at x = 0.6 and 0.8, the trends for the p profiles are in reverse to the data (i.e., decreased and increased values, respectively), whereas at x = 0.9, the agreement with the data is good (i.e., reduced values). These differences are likely due to the differences in the calculated and measured afterbody shoulder-wave trough. The Fr differences for the velocity and pressure profiles penetrate to larger depths than on the forebody, as was also the case with the data.

Lastly, at x = 1, 1.1, and 1.2, the H contours show a reduction in both the depthwise and width extents of the wake, whereas the data only displayed the former. The u contours are similar, but also display a broad region of u < 1 that shifts outward with increasing x due to the effects of the stern wave, although reduced in magnitude and extent in comparison to the data, no doubt, due to the underprediction of the stern-wave amplitudes. The v-w display similar trends to the data (i.e., at x = 1 and 1.2, increased upward and downward flow, respectively, and a stronger longitudinal vortex that shifts outward); however, the longitudinal vortex is not well defined. The p values are larger than those for the data (i.e., increased values due to the stern-wave crest, especially at x = 1 and 1.2, except in the longitudinal vortex core). The trends for the ω_x contours are similar to the data (i.e., display larger values, especially near the free surface, and the shift of the longitudinal vortex core). The k contours continue to correlate with H. The H profiles display larger values for Fr = 0.316, as did the data for the outer, but not the inner region. At x = 1 and 1.2, the u profiles indicate larger values for Fr = 0.316, but with smaller differences than indicated by the data. At x = 1.1 the calculated u profiles display minimal Fr effects, whereas the data indicated a slight reduction in magnitude. At x = 1 and 1.1, the v profiles show larger values, and at x = 1.2, smaller and larger values for the inner and outer parts of the profiles, respectively, which is similar to the data and correlates with ζ_{y} . The w profiles show larger and smaller values at x = 1and 1.1, 1.2, respectively, which is similar to the data and correlates with ζ_x . In general, the v and w profiles show less differences than the data and lack detail. The p profiles show qualitatively similar trends (i.e., increased values), but larger magnitudes, than the data.

In summary, the calculations show overall close agreement to the data, including both qualitative (i.e., most of the trends are accurately predicted) and quantitative (i.e., most of the velocity and pressure magnitudes are accurately predicted) similarity; however, as will be discussed further later, there are certain systematic discrepancies. In particular, the complex topology of the wave field is not resolved, which is attributed to deficiencies of the inviscid-flow method, and the very detailed viscous flow, especially near the hull surface and wake centerplane, is not resolved, which is attributed to deficiencies associated with the use of wall functions and the current limitations of cfd.

Comparison with inviscid-flow results

As mentioned earlier, the inviscid-flow results were previously evaluated in [13] through detailed comparisons with the present data. In summary, the inviscid-flow calculations in many respects, showed remarkably close agreement with the data, i.e., the general features of the wave system and velocity and pressure fields in the outer region were fairly accurately simulated. However, the

complex details of the wave field were not predicted and viscous flow close to the hull and wake centerplane was completely absent. With regard to the former, the amplitude of the bow-wave system was underpredicted, there were differences in both amplitude and phase for the shoulderwave system, and the amplitudes of the stern waves were overpredicted and lack complexity as was also the case with the global-region wave system. With regard to the latter, there were large differences in the detailed flow close to the hull and wake centerplane, including the stagnation effects at the bow, the bilge vortices on the hull and longitudinal vortices in the wake were absent, as was the complex flow on the afterbody and in the wake.

The interactive-approach calculations indicate significant improvements with regard to both of the aforementioned shortcomings of the inviscid calculations, i.e., inclusion of displacement effects in the inviscid calculations reduces the pressure rise at the stern and the amplitudes of the stern waves resulting in better agreement with the data; and the viscous-flow calculations, including the wave-induced pressure gradients, enable an accurate simulation of the viscous flow close to the hull and wake centerplane.

WAVE/BOUNDARY-LAYER AND WAKE INTERACTION

The foregoing discussions indicate pronounced wave-induced effects as well as the complexity of the interaction between the wavemaking and boundary layer and wake. In the following, the results are discussed further to assess the nature of this interaction.

Very significant differences are observed between the Fr = 0 and 0.316 results due to wave-induced effects. On the forebody ($0 \le x \le 0.4$), the differences are due to the bow-wave crest and forebody shoulder-wave trough, and are largest near the free surface and decay with depth. Initially, the former alters the stagnation effects at the bow and induces a broad region of reduced axial velocity and turns the crossplane flow upwards and away from the hull and then downwards and towards and away from the hull in the inner and outer regions, respectively. Subsequently, the latter induces increased axial velocity and downward crossplane flow. Although the boundary layer shows some thinning, the differences are relatively small and the bow-bilge vortex is apparently unaltered, both of which are surprising in consideration of the large differences in pressure gradients for high and low Fr. Apparently, the boundary layer is sufficiently thin such that it is relatively unresponsive to the changes in pressure gradients.

On the afterbody $(0.6 \le x \le 1)$ and in the near wake $(1 \le x \le 1.2)$, the differences are due to the shoulder-wave system (i.e., the crest between the shoulder-wave troughs and the afterbody shoulder-wave trough) and the stern-wave crest and, here again, are largest near the free surface and decay with depth, but with more significant effects at larger depths than was the case on the forebody. The crest and trough wave effects on the axial velocity and crossplane flow in the outer region are similar as described for the forebody. The crossplane flow displays significantly increased magnitudes. Very importantly, in this case, the differences within the boundary layer are appreciable. The boundary layer and wake thickness is reduced substantially, especially near the free surface. The stern-bilge vortex on the hull and longitudinal vortex in the wake are larger and altered in position. Interestingly, the pressure recovery on the hull at the stern is reduced; however, the pressure is larger in the near wake due to the effects of the stern-wave crest. In many cases, the shape of the detailed velocity and pressure profiles are completely altered.

In most cases, the differences between the flows for high and low Fr can be directly related to the wave elevations and wave-induced pressure gradients. The differences for the *u* and *w* velocity components were shown to be correlated with ζ and ζ_x , respectively, and those for the *v* velocity component to be consistent with ζ_y . In some cases, the differences for *u* and *w* were shown to be consistent with the variations in p_x and p_z , respectively. The *H* and *p* differences usually were correlated with those for *u*. It should be recognized that the fact that *u* and *w* can be correlated with ζ and ζ_x , respectively, is consistent with the in-phase relationship between these quantities for a Stokes wave. Of course, their behavior is also related to other quantities, e.g., the behavior of *u* is certainly related to ζ_x ; however, the response is out-of phase and more difficult to identify. Here again, this is consistent with the fact that *u* and ζ_x for a Stokes wave are 90° out-of-phase. It should be mentioned that other factors undoubtedly have important influences, e.g.,

CAF 23/6-E

wave-induced separation, breaking waves, and unsteady effects, all of which played a relatively small role in the present application.

As discussed earlier, the data and the inviscid-flow calculations also indicate very significant differences. Most importantly, the inviscid solution lacks "real-fluid effects," i.e., the viscous flow close to the hull and wake centerplane is clearly not accurately resolved, and, in addition, the complex details of the wave field are not predicted. Also, the bare-body results clearly overpredict the pressure recovery at the stern and the stern-wave amplitudes. The interactive results show improvement with regard to both of these shortcomings.

In conclusion, the wave-boundary layer and wake interaction is most significant at the bow and on the afterbody and in the wake. In these areas, both the viscous and inviscid flows are altered due to their interactions. In the latter case, most of the interaction can be explicated as a result of the wave elevations, wave-induced pressure gradients, and the displacement effects of the boundary layer.

CONCLUDING REMARKS

An interactive approach for calculating ship boundary layers and wakes for nonzero Froude number has been validated through detailed comparisons with recent extensive experimental data for the practical 3-D geometry of the Series $60 C_B = 0.6$ ship model at both low (0.16) and high (0.316) Froude number. Close agreement is demonstrated between the calculations and the data, which supports the conclusion that the present approach can accurately predict ship boundary layers and wakes, including free-surface effects; however, the present detailed comparison enables a more critical evaluation, which will be given subsequently. Additionally, comparisons are made with inviscid-flow results, which in combination enable an evaluation of the wave/boundary-layer and wake interaction. On the forebody, the differences are primarily in the outer (inviscid) flow, except at the bow, whereas on the afterbody and in the near wake, both the inner (viscous) and outer flows are altered. Most of the interaction between wavemaking and the boundary layer and wake can be explicated as a result of the wave elevations, wave-induced pressure gradients, and the displacement effects of the boundary layer.

The interactive approach utilizes the leading inviscid- and viscous-flow technologies, including their interactions; thereby, reflects both their current capabilities and limitations. Inclusion of displacement effects in the inviscid calculations surely improves the predictions with regard to certain features (i.e., the pressure rise at the stern and the stern-wave amplitudes), but cannot significantly alter the overall limitation of potential-flow based methods in simulating the complex details of the wave field, including an accurate prediction of the wave slopes. Although nonlinear free-surface boundary conditions can and should be included, it seems unlikely that this will significantly improve the detailed predictions. This statement is supported by the recent results from extensions of an inviscid-flow method similar to the present one to include nonlinear free-surface boundary conditions [22]. Similarly, inclusion of the wave-induced pressure gradients in the viscous-flow solution enables the calculation of the boundary layer and wake for nonzero Fr, but cannot alter the overall current limitations in cfd, which were discussed earlier with regard to the Fr = 0 results. In conclusion, the interactive approach is limited by the inherent limitation of the inviscid technology and current pacesetting issues in cfd development. This is considered an overall assessment of the interactive approach; since, although alternative interaction procedures (e.g., [10, 11]) may offer some advantages over the present rather straightforward displacement-body approach (e.g., in extensions for full-form ships), they cannot alter the limitations of the inviscid and viscous technologies.

Although recent advancements in the development of large domain approaches [5-8] have led to certain improvements in their capabilities, it is still apparent, as stated in [9], that the present results appear superior both with regard to the resolution of the boundary layer and wake and the wave field. Furthermore, it appears that the present interactive approach is considerably more computationally efficient than the large-domain approaches of [5-8]. This is consistent with the previous finding for zero Fr [18]; however, a complete evaluation requires a detailed comparative study.

As mentioned in the Introduction, a large-domain approach is currently under development both

for the purpose of performing such an evaluative comparison and due to the future promise of this approach for a more detailed resolution of the entire flow albeit most likely computationally more intensive. The large-domain approach is essentially a "complete" viscous-flow approach and thus its limitations are those for cfd with some additional issues associated with the free-surface boundary conditions. The latter of which are required for detailed resolution of the flow very close to the free surface and, most likely, the complex topology of the wave field. The status of the development includes extensions and modifications for many of the earlier mentioned advancements in viscous-flow methodologies in conjunction with the utilization of free-surface conforming grids and inviscid free-surface boundary conditions. The results thus far indicate improved resolution of the boundary layer and wake due to the use of near-wall turbulence modeling and the wave profile and local region wave field due to the use of free-surface conforming grids, but difficulties in detailed resolution of the global region wave field due to limitations in sufficient grid density over the entire domain [24]. Future work involves a more detailed comparative study, including an assessment of the relative influences of nonlinear and viscous effects. Also, recent progress on the solution of a solid-fluid juncture boundary layer and wake with waves, including exact and approximate free-surface boundary conditions for laminar and turbulent flow, respectively, offers encouragement in extending these techniques for the present application [25]. The work concerning the large-domain approach will be reported in a future publication.

The present work has several implications with regard to surface-ship design: the wave pattern, breaking, and -induced separations along with turbulence/vortex/free-surface interaction, bubble entrainment, etc. are key issues with regard to performance prediction, signature reduction, and propeller-hull interaction. Note that it is a straightforward extension of the present approach to include propeller-hull interaction using the approach of [4]. In the near term, the interactive approach offers advantage over the large-domain approach both with regard to computational accuracy and efficiency and thus is recommended for use in design, but restricted to relative performance evaluation and design trade-off studies due to current limitations, as described earlier. A similar conclusion was reached in [4] with regard to the capabilities of cfd for predicting propeller-hull interaction. In general, this is the current situation for cfd not only for the present applications, but for others, e.g., in the aerospace industry. In the long term, the large-domain approach offers the possibility of a more detailed resolution of the overall flow field and, therefore, will ultimately replace the interactive approach.

Acknowledgements-This research was sponsored by the Office of Naval Research under Grant N00014-92-J-1092 under the administration of Dr E. P. Rood whose support and helpful technical discussions are greatly appreciated.

REFERENCES

- 1. L. Larsson, V. C. Patel and G. Dyne, Ship Viscous Flow: Proceedings of 1990 SSPA-CTH-IIHR Workshop. Research Report No. 2, Flowtech International AB, Gothenburg (June 1991).
- 2. J. Piquet and M. Visonneau, Computation of the flow past shiplike hulls. Computers Fluids 19, 183 (1991).
- 3. C.-W. Lin, G. D. Smith and S. C. Fisher, Numerical viscous flow simulation of wind tunnel model-scaled experiments. Proc. 2nd Osaka International Colloquium on Viscous Fluid Dynamics in Ship and Ocean Technology, Osaka, Japan, pp. 251-270 (September 1991).
- 4. F. Stern, H. T. Kim, D. H. Zhang, Y. Toda, J. Kerwin and S. Jessup, Computation of viscous flow around propeller-body configurations: Series 60 $C_{\rm B} = 0.6$ Ship Model. J. Ship Res. 38, 136 (1994).
- 5. H. Miyata, M. Zhu and O. Watanabe, Numerical study on a viscous flow with free-surface waves about a ship in steady straight course by a finite-volume method. J. Ship Res. 36, 332 (1992).
- 6. T. Hino, Computation of viscous flows with free surface around an advancing ship. Proc. 2nd Osaka Int. Colloquium on Viscous Fluid Dynamics in Ship and Ocean Technology, Osaka, Japan, pp. 83-101 (September 1991).
- 7. S.-H. Kwag, K.-H. Mori and Y. Doi, Numerical simulation of high Reynolds number flows with free surface. Proc. 18th ONR Symp. on Naval Hydro, Ann Arbor, Mich. (1990).
- 8. J. Farmer, L. Martinelli and A. Jameson, A fast multigrid method for solving the nonlinear ship wave problem with a free surface. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).
- Y. Tahara, F. Stern and B. Rosen, An interactive approach for calculating shp boundary layers and wakes for nonzero Froude number. J. Comput. Phys. 98, 33 (January 1992). 10. H. C. Chen, W. M. Lin and K. M. Weems, Interactive zonal approach for ship flow including viscous nonlinear and
- wave effects. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).
- 11. E. Campana, A. Di Mascio, P. G. Esposito and F. Lalli, Domain decomposition in free surface viscous flows. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).
- 12. Y. Toda, F. Stern and J. Longo, Mean-flow measurements in the boundary layer and wake and wave field of a Series $60 C_B = 0.6$ Ship model—Part 1: Froude numbers 0.16 and 0.316. J. Ship Res. 37, 360 (December 1992).

Y. TAHARA and F. STERN

J. Longo, F. Stern and Y. Toda, Mean-flow measurements in the boundary layer and wake and wave field of a Series 60 C_B = 0.6 Ship model—Part 2: Scale effects on near-field wave patterns and comparisons with inviscid theory. J. Ship Res. 37, 16 (March 1993).

14. B. Rosen, SPLASH free-surface code: theoretical/numerical formulation. South Bay Simulations Inc., Babylon, NY (1989).

- 15. V. C. Patel, H. C. Chen and S. Ju, Ship stern and wake flows: solutions of the fully-elliptic Reynolds-averaged Navier-Stokes equations and comparisons with experiments. Iowa Institute of Hydraulic Research, The University of Iowa, IIHR Report No. 323 (1988); also J. Comput. Phys. 88, 305 (June 1990).
- 16. B. Maskew, Prediction of subsonic aerodynamic characteristics: a case for low-order panel methods. J. Aircraft 19, 157 (1982).
- 17. B. Maskew, A computer program for calculating the non-linear aerodynamic characteristics of arbitrary configurations. NASA CR-166476 (1982).
- 18. F. Stern, S. Y. Yoo and V. C. Patel, Interactive and large domain solutions of higher-order viscous-flow equations. AIAA J. 26, 1052 (1988).
- F. Stern, D. H. Zhang, B. Chen, H. T. Kim and S. Jessup, Computation of viscous marine propulsor blade and wake flow. Proc. 20th ONR Symp. on Naval Hydro., Santa Barbara, CA (August 1994).
 B. Chen, F. Stern and W. J. Kim, Computation of unsteady viscous marine propulsor blade and wake flow. Proc. 20th
- 20. B. Chen, F. Stern and W. J. Kim, Computation of unsteady viscous marine propulsor blade and wake flow. Proc. 20th ONR Symp. on Naval Hydro., Santa Barbara, CA (August 1994).
- 21. E. P. Paterson and F. Stern, Computation of unsteady viscous flow with application to the MIT flapping-foil experiment. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).
- 22. H. C. Raven, Nonlinear ship wave calculations using the RAPID method. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).
- ITTC, Report of the resistance and flow committee. Proc. 18th Int. Towing Tank Conf., Kobe, Japan, pp. 47–92 (1987).
 Y. Tahara and F. Stern, A large-domain approach for calculating ship boundary layers and wakes for nonzero Froude
- numbers. Proc. CFD Workshop Tokyo 1994, Tokyo, Japan, Vol 1, pp. 45-55 (March 1994).
 25. J.-E. Choi and F. Stern, Solid-fluid juncture boundary layer and wake with waves. Proc. 6th Int. Conf. on Numerical Ship Hydrodynamics, Iowa City, IA (August 1993).

An Application of Two-Layer k- ε Model to Ship Flow Computation

by Yusuke Tahara*, Member

Summary

A numerical method is developed for calculating ship boundary layers and wakes for zero Froude number (i. e., the free surface is considered to be flat and treated as a plane of symmetry). The Reynolds-averaged Navier-Stokes and continuity equations are solved with the two-layer $k-\varepsilon$ turbulence model. The results are validated through comparisons with the experimental data for the Series $60C_B = .6$ ship model and results of the precursory wall function as well as the Baldwin-Lomax model approaches. The three turbulence models yield satisfactory results; however, the present and Baldwin -Lomax model results indicate improved resolution of the flow close to the hull and wake centerplane due to near-wall turbulence modeling. An overview is given of the present approach, and a discussion is provided of the relative performance of three turbulence models. Lastly, some concluding remarks are made, including the requirements and prognosis for improvements.

1. Introduction

Considerable effort has been put forth in the development of numerical methods to calculate ship boundary layer and wake flows. Many studies have focused on the development of Reynolds-averaged Navier-Stokes (RaNS) equation methods with a variety of turbulence models^{1),2)}. However, the studies were, in general, not systematic with regard to documentation and isolation of the specific influences of computational methods and turbulence models. The present study is concerned with an application of the two-layer $k-\varepsilon$ model to ship flow calculation, and detailed evaluation of different turbulence models with the same numerical scheme.

In the RaNS equation method, a turbulence model must be introduced to effect the closure. Quite a few turbulence models have been proposed. An algebraic eddy-viscosity model of Baldwin and Lomax is very popular because of its simplicity. For example, Tahara and Stern³⁾ used this model in the recent study, and showed that considerably accurate solutions were obtained not only for zero Froude number (Fr) but for nonzero Fr. However, application of this model is limited to simple flow field. As seen in the recent workshop¹⁾, this model in nonmodified form generally failed to reproduce complicated flows around a tanker ship.

On the other hand, one or two equation turbulence

* University of Osaka Prefecture

Received 9th Jan. 1995

Read at the Spring meeting 17, 18th May 1995

models are also popular and have been studied by many researchers. For instance, Patel et al.⁴⁾, and Tahara et al.^{5),6),7)} applied the k- ε model to ship flow computation with a two-point wall-function approach, and showed that many important features of the flow were adequately simulated. It appeared that the wall-function approach increased the computational efficiency; however, some difficulties were also observed, such that the flows very close to the hull or in the wake region were not accurately reproduced.

A modification to the method⁴) was proposed by Chen and Patel⁸⁾. In the study, a generalized version of the two-layer approach was introduced, in place of the wallfunction approach. Using the approach, Chen and Patel⁸⁾ calculated the flow around wing-body junctions, and Kim⁹⁾ and Kim¹⁰⁾ investigated flow separation on a spheroid at incidence, and longitudinal vortices in turbulent boundary layers, respectively. As shown in the work, the two-layer $k - \varepsilon$ model is very useful to study complicated flows involving separation, and can replace the wall function through relatively simple modification.

In the present study a numerical method is developed, which is based on extensions of the precursory wall-function approach of Tahara⁷, including near-wall turbulence modeling of the two-layer $k-\epsilon$ model. The results are validated through comparisons with the experimental data for the Series $60C_B=.6$ ship model^{11),12} and results of the precursory wall-function⁷ and Baldwin-Lomax model³ approaches. In the following, the computational method, conditions, and grid are described and results discussed with regard to the experimental data and relative accuracy of three turbulence models. Lastly, some concluding remarks are made, including the requirements and prognosis for improvements.

2. Governing equation

The non-dimensional RaNS equations for unsteady, three-dimensional incompressible flow can be written in Cartesian tensor notation as

$$\sum_{i=1}^{3} \frac{\partial U_{i}}{\partial x^{i}} = 0$$

$$(1)$$

$$\frac{\partial U_{i}}{\partial t} + \sum_{j=1}^{3} \left(U_{j} \frac{\partial U_{i}}{\partial x^{j}} + \frac{\partial \overline{u_{i}u_{j}}}{\partial x^{j}} \right) + \frac{\partial p}{\partial x^{i}} - \frac{1}{Re} \nabla^{2} U_{i} = 0$$

$$(2.a)$$

with

$$\nabla^2 = \sum_{j=1}^3 \frac{\partial^2}{\partial x^j \partial x^j} \tag{2 b}$$

where $U_i = (U, V, W)$ and $u_i = (u, v, w)$ are the Cartesian components of mean and fluctuating velocities, respectively, normalized by the reference velocity U_0 , $x^i = (x, y, z)$ are the dimensionless coordinates normalized by a characteristic length L, p is the pressure normalized by ρU_0^2 , $Re = U_0 L/\nu$ is the Reynolds number, ν is the kinematic viscosity, and the barred quantities $-\overline{u_i u_j}$ are the Reynolds stresses normalized by U_0^2 .

If the Reynolds stresses $-\overline{u_iu_j}$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity ν_t , i. e.

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i} \right) - \frac{2}{3} \delta_{ij} k \qquad (3)$$

where $k = (\overline{uu} + \overline{vv} + \overline{ww})/2$ is the turbulent kinetic energy, equation (2) for the mean momentum becomes

$$\frac{\partial U_i}{\partial t} + \sum_{j=1}^{3} \left[\left(U_j - \frac{\partial v_t}{\partial x^j} \right) \frac{\partial U_i}{\partial x^j} - \frac{\partial v_t}{\partial x^j} \frac{\partial U_j}{\partial x^i} \right] \\ + \frac{\partial}{\partial x^i} \left(p + \frac{2}{3}k \right) - \frac{1}{R} \nabla^2 U_i = 0$$
(4)

where $1/R = 1/Re + \nu_t$.

Equations (1) and (4) can be solved for U_i and p when a suitable turbulence model is employed to calculate the eddy-viscosity distribution. In this study, eddy viscosity is given by the two-layer model of Chen and Patel³⁾. In the outer layer, eddy viscosity is defined by $\nu_t = C_{\mu}k^2/\varepsilon$, where ε is the rate of turbulent energy dissipation. k and ε are obtained from the transport equations

$$\frac{\partial k}{\partial t} + \sum_{j=1}^{3} \left(U_{j} - \frac{1}{\sigma_{k}} \frac{\partial \nu_{t}}{\partial x^{j}} \right) \frac{\partial k}{\partial x^{j}} - \frac{1}{R_{k}} \nabla^{2} k$$

$$-G + \varepsilon = 0 \qquad (5 a)$$

$$\frac{\partial \varepsilon}{\partial t} + \sum_{j=1}^{3} \left(U_{j} - \frac{1}{\sigma_{\epsilon}} \frac{\partial \nu_{t}}{\partial x^{j}} \right) \frac{\partial \varepsilon}{\partial x^{j}} - \frac{1}{R_{\epsilon}} \nabla^{2} \varepsilon$$

$$-C_{\varepsilon_{1}} \frac{\varepsilon}{k} G + C_{\varepsilon_{2}} \frac{\varepsilon^{2}}{k} = 0 \qquad (5 b)$$

where R_k and R_{ε} are the effective Reynolds numbers defined by $1/R_{\phi}=1/Re+\nu_t/\sigma_{\phi}$, $\phi=k, \varepsilon$, and

$$G = \frac{1}{2} \nu_t \sum_{i=1}^3 \sum_{j=1}^3 \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i} \right)^2$$
(5 c)

is the rate of production of k, and $(C_{\mu}, C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_{k}, \sigma_{\varepsilon})$ are constants whose values are (0.09, 1.44, 1.92, 1.01, 1.3).

In the near-wall layer, only k is obtained from the

transport equation (5 a), then ν_t and ϵ are given by ν_t $=C_{\mu}\sqrt{k} l_{\mu}$ and $\varepsilon = k^{3/2}/l_{\epsilon}$, where l_{μ} and l_{ϵ} are the length scales defined by $l_{\mu} = C_{l} Y_{n} [1 - \exp(-R_{y}/A_{\mu})]$ and $l_{\epsilon} =$ $C_t Y_n [1 - \exp(-R_y/A_\epsilon)]$. $R_y = Re\sqrt{k} Y_n$ is the turbulent Reynolds number and Y_n is the distance from the wall. $C_l = \kappa C_{\mu}^{-3/4}$, $A_{\mu} = 70$, and $A_{\epsilon} = 2C_l$ are constants used by Chen and Patel⁸⁾, where $\kappa = 0.418$ is the von Karman constant. The value of C_i was determined to ensure a smooth eddy-viscosity distribution at the boundary of the near-wall and outer layers. A_{μ} was determined from numerical tests so as to recover the additive constant B=5.45 in the logarithmic law in the case of a flat plate boundary layer. The value of A_{ϵ} was assigned so as to recover the proper asymptotic behavior $\varepsilon =$ $2\nu k/Y_n^2$ in the sublayer. Note that the near-wall layer includes the laminar sublayer, the buffer layer, and a part of the fully-turbulent logarithmic layer.

3. Discretization and velocity-pressure coupling

The five transport equations (4) and (5) are discretized by the finite-analytic scheme. In the scheme, equations are first linearlized in each local numerical element, then solved analytically by the method of separation of variables. Evaluation of the analytic solutions at the interior node then provides a twelvepoint discretization formula of the form of

$$\phi_{P} = \frac{1}{1 + C_{P} \left[C_{U} + C_{D} + \frac{R}{\Delta \tau} \right]} \left\{ C_{NE} \phi_{NE} + C_{NW} \phi_{NW} + C_{SE} \phi_{SE} + C_{SW} \phi_{SW} + C_{EC} \phi_{EC} + C_{WC} \phi_{WC} + C_{NC} \phi_{NC} + C_{SC} \phi_{SC} + C_{P} \left(C_{U} \phi_{U} + C_{D} \phi_{D} + \frac{R}{\Delta \tau} \phi_{P}^{n-1} \right) - C_{P} (S_{\phi})_{P} \right\}$$

$$(6)$$

The solution of the complete flow equations involves a global iteration process, in which the velocity-pressure coupling is effected by predictor-corrector steps. In the predictor step, the pressure field at the previous time step is used in the solution of the implicit equations to obtain the corresponding velocity field. Since the velocity field generally does not satisfy mass conservation, a corrector step is needed. In the corrector step, the momentum equations and the implicit pressure equation are solved iterativly to ensure the satisfaction of the continuity equation. For more detailed equations or references, see Tahara⁷¹ or Chen and Patel⁸⁰. The present overall solution procedure follows Tahara⁷¹, which this study is based on.

4. Boundary and initial conditions

Consider a ship fixed in the uniform onset flow $\vec{U}_0 = (U_0, 0, 0)$ as depicted in Fig. 1. Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward. The solution domain is shown in Fig. 2. In terms of the notation of Fig. 2, the boundary conditions on each of the boundaries are described in the following.



Fig. 1 Coordinate system



Fig. 2 Solution domain and boundaries

On the inlet plane Si, the boundary conditions are provided on the basis of the freestream values, i. e., U=1, $V = W = \partial(k, \epsilon)/\partial x = p = 0$; on the exit plane Se, axial diffusion is negligible so that the exit conditions used are $\partial^2(U, V, W, k, \varepsilon)/\partial x^2 = \partial p/\partial x = 0$; on the symmetric plane Sc, the conditions imposed are $\partial(U, W, k)$ ϵ , $p)/\partial y = V = 0$; boundary conditions imposed on the waterplane Sw are similar to those on Sc, such that $\partial(U, V, k, \varepsilon, p)/\partial z = W = 0$; and on the outer boundary So, the conditions imposed are U=1, $W=\partial(k, \varepsilon)/\partial r=p$ =0 (where r is the radial direction as shown in Fig. 2), and V is determined by the solution to satisfy the continuity equation. On the body surface Sb, boundary conditions used are $U = V = W = k = \partial p / \partial n = 0$ (where n is normal to the body). On Sb, the boundary condition for ε is unnecessary because ε in the near-wall layer is expressed algebraically, which gives the boundary condition for (5 b) in the outer layer.

In this study, the initial conditions are taken from the freestream values, i. e., U=1, V=W=p=0; and very small values are specified for k and ϵ . It is assumed that the flow is already turbulent upstream of the bow.

5. Ship geometry and experimental information

As already indicated, the computational results are presented for the Series $60C_B = .6$ ship model for which an extensive set of experimental data were recently obtained by Toda et al.¹¹ at the Iowa Institute of

Hydraulic Research (IIHR) towing tank. Mean-velocity and pressure field measurements were performed for $Fr = U_0 \sqrt{gL} = 0.16$ (average Re = 2,220,000) and 0.316 (average Re = 4,340,000) for a 3.048 m model at numerous stations from the bow to the stern and into the near wake. The principal dimensions and offsets of the model are given in Toda et al.¹¹). In the present study, the computational results are compared with the low Frexperimental data, which are referred to as IIHR in the following.

In addition, more experimental data especially near the stern and wake region are available¹²⁾. The measurements were performed at the Osaka University towing tank for 4 m models for Fr=0.16 and Re=3,200, 000. These data, which are referred to as OSAKA, are also used for comparison.

6. Results and discussion

In the following, first the computational grids and conditions are described. Then results are presented and discussed, including wherever possible comparisons with available experimental data. In the presentation of the results, variables are non-dimensionalized using the ship length L, the free-stream speed U_0 and fluid density ρ .

The computational grids were obtained using the elliptic-algebraic method³⁾. The inlet, exit and outer boundaries are located at x=(-0.4, 2.0) and r=1, respectively. The first grid point off the body surface is located in the viscous sublayer, i. e., $y^+ < 0.2$, where $y^+ =$ $ReU_{\tau}Y_n = Re\sqrt{\tau_w/\rho U_0^2} Yn$ is the dimensionless distance, and τ_w is the wall-shear stress. The matching boundary between the near-wall and outer layers is located in 50 $< y^{+} < 100$, where the location was initially guessed, then updated several times until the requirement was satisfied. The numbers of grids used are $90 \times 60 \times 20$ (= 108,000) in the axial, radial, and circumferential directions, respectively. The conditions for the computation were as follows: L=1; $U_0=1$; Re=2,000,000. The values of the time increment Δr and underrelaxation factors for velocity $(\alpha_{U}, \alpha_{V}, \alpha_{W})$, turbulence quantities $(\alpha_k, \alpha_{\epsilon})$ and pressure (α_p) were as follows: $\Delta \tau = 0.01$; $\alpha_{v} = \alpha_{v} = \alpha_{w} = 1$; $\alpha_{k} = \alpha_{e} = 0.01$; and $\alpha_{p} = 0.1$. The supercomputer (SX-3 R) time and central memory were about 1 hour and 60 MB. The convergence criterion was that the change in solution be less than about 0.05%for all variables, which was satisfied in 800 global iterations.

Figs. 3 through 8 show the axial-velocity (U) contours, crossplane vectors (V-W), and pressure (Cp = p/.5) contours for several representative sections, i.e., x = (0, 0.2, 0.4, 0.8, 1.0, 1.1). Figs. 9 through 14 provide detailed comparisons of the velocity and pressure profiles between the computations and experimental data¹¹ (referred to as IIHR) at the same stations. In Figs. 12 through 14, the measurements¹² (referred to as OSAKA) are also included. Reynolds stress contours,

Journal of The Society of Naval Architects of Japan, Vol. 177



Fig. 3 Comparison of solutions at x=0: (a) Axialvelocity contours; (b) Crossplane-velocity vectors; and (c) Cp contours



velocity contours; (b) Crossplane - velocity vectors; and (c) C_P contours



Fig. 5 Comparison of solutions at x=0.4: (a) Axialvelocity contours; (b) Crossplane-velocity vectors; and (c) C_p contours













-111 -

near-wall velocity profiles, and surface pressure and shear stress distributions are compared in Figs. 15 through 21 between the two-layer and Baldwin-Lomax (B-L) model results. Lastly, Table. 1 and Fig. 22 provide comparison of resistance coefficients between the computations and experimental data. Those results are discussed in the following.

Figs. 3 and 9 show comparisons of results at x=0. At this station, the experimental data display that the flow is dominated by stagnation effects associated with the close proximity of the bow. The same trends of the Uand C_P contours are observed in the present computation (i.e., the U contours are less than unity through the extent of the present region; and the C_P contours are similar to those for U, but with reverse trend in magnitude). The computed V-W vectors show the double model stagnation flows; however, the measurements show downward shift of the stagnation point due to bow-wave effects. The trends of the U and V as well as C_p profiles of the measurements are well reproduced by the computations (i.e., the U profiles display the appearance of laminar stagnation-point flow; the V profiles are positive at all depths due to outward displacement effects of the hull; and the C_{P} profiles display quite large values in the inner part). The measured W profiles are upward near the free surface and downward at larger depths; but the computations are downward at all depths due to the aforementioned reason. No significant differences are observed between the two-layer and B-L model results, which show closer agreement with the measurements than the wall-function results.

Figs. 4 and 10 show comparisons of results at x=0.2. At this station, similar trends of the U and C_P contours are observed between the measurements and present computation (i. e., the U contours display the thickening of the boundary layer, especially near the keel, which is due to the flow convergence toward the keel; and the C_P contours show the lowest values near the hull, especially in the region near the bow-bilge vortex). The computed and measured V-W vectors show many similarities of complex flow features due to the bow-bilge vortex (i. e., the V-W vectors mostly show downward flow, and an increased magnitude can be observed near the bilge; and upward flow near the centerplane is observed); but the present computation did not improve one of the previous issues⁷, i.e., the vectors have lower estimated magnitude near the bilge. The U profiles of the two-layer and B-L model results show clear improvement over the wall-function results especially near the hull. However, at the largest depth, the two-layer results show slightly thicker boundary layer than the B-L model results. The V and W as well as C_{P} profiles are also improved in the two-layer and B -L model results, but lack details very close to the hull. Figs. 5 and 11 show comparisons of results at x=0.4.

At this station, trends of the computed U contours are

very similar to those of the measurements (i. e., the U contours show the continued growth of the boundary layer and the convergence of the flow toward the keel, i. e., the boundary layer is quite thick near the keel, thin near the bilge, and somewhat thicker near the free surface). This is also true for the V-W vectors and C_P contours (i. e., the V-W vectors have decreased magnitude except near the bilge where the remnants of the bow-bilge vortex exist; and the C_P contours display decreasing values from x=0.2 to the present section, except in the region near the bow-bilge vortex). The trends displayed in the velocity and pressure profiles are consistent with the above. No significant differences are observed among three computations except for the V profiles very close to the hull at the largest depth.

Figs. 6 and 12 show comparisons of results at x=0.8. At this station, the same trends are observed between the measured and computed U contours (i.e., the Ucontours display a pronounced bulge in the boundary layer near the region of maximum hull concavity and reduction in the boundary layer near the centerplane). The present computation shows clear improvement over the previous wall-function results. The V-Wvectors and C_P contours are also very similar to those of the measurements (i. e., the crossplane vectors are directed upward and toward the centerplane, in which the presence of a stern-bilge vortex is evident; and the C_P contours show negative values everywhere, with the lowest values in the region near the stern-bilge vortex). The U profiles of the two-layer and B-L model results show good agreement with the measurements (i. e., the U profiles display the detailed characteristics of the thick boundary layer in this section), and indicate clear improvement over the previous wall-function results; however at the larger depths, the two-layer model somewhat overpredicts the boundary layer thickness. The V and W profiles of the computations also show good agreement with the measurements, but lack details very close to the hull. The C_P profiles of the measurements are well reproduced in the computations, where no significant differences are observed among three turbulence model results.

Figs. 7, 8, 13, and 14 show comparisons of results at x = 1.0 and 1.1. At these stations, the measurements display merging of the boundary layer into the wake and its initial evolution. Those are clearly observed in the computed U contours, where the measurements show a relatively slower recovery near the free surface, which may be due to wave effects. The present U contours show more complexities than the previous wall-function results, and are more similar to those of the measurements. The present V-W vectors show the same trends as those of the measurements (i. e., the flow is upward and toward the centerplane, in which the stern-bilge vortex evolves into a weak longitudinal vortex), and clear improvements over the previous wall-function results are observed. The present C_P contours

-112 -



Fig. 9 Velocity and pressure profiles at x=0



Fig. 10 Velocity and pressure profiles at x=0.2



Fig. 11 Velocity and pressure profiles at x=0.4

-115-





170

-116-







Fig. 14 Velocity and pressure profiles at x=1.1

are also similar to those of the measurements, but still lack details. The above is more clearly displayed in the velocity and pressure profiles. The U profiles of the two-layer and B-L model results show clear improvement over the wall-function results; however at x=1.1, some discrepancies are observed between the two-layer model results and the measurements at larger depths. The V and W profiles are also improved in the twolayer and B-L model results, where the W profiles of the two-layer results show the closest agreement with the measurements. The C_P profiles of the computations show good agreement with the measurements, where the two-layer and B-L model results are closer to the experimental data than the wall-function results.

Figs. 15 through 17 provide comparison of Reynolds stresses $-\overline{uv}$, $-\overline{uw}$ and $-\overline{vw}$, between the two-layer and B-L model results at several representative stations, i. e., x = (0.4, 0.8, 1.1). Note that the two results are presented for the starboard side of ship, i. e., y > 0. Because both turbulence models follow the Boussinesq assumption and the isotropic eddy viscosity, the general features of Reynolds stresses can be correlated with mean velocity gradient, such that $-\overline{uv}$ and $-\overline{uw}$ are mainly related to $\partial U/\partial y$ and $\partial U/\partial z$, respectively. On the other hand, more complicated influences may exist between $-\overline{vw}$ and $\partial W/\partial y$ and $\partial V/\partial z$, which are related to the secondary motion of the flow, e.g., the longitudi-



Fig. 15 Comparison of Reynolds stresses at x=0.4: (a) $-\overline{uv}$ contours; (b) $-\overline{uw}$ contours; (c) $-\overline{vw}$ contours; columns, B-L model and two-layer k- ε model, respectively



Fig. 16 Comparison of Reynolds stresses at x=0.8: (a) $-\overline{uv}$ contours; (b) $-\overline{uw}$ contours; (c) $-\overline{vw}$ contours; columns, B-L model and two-layer $k-\varepsilon$ model, respectively



Fig. 17 Comparison of Reynolds stresses at x=1.1: (a) $-\overline{uv}$ contours; (b) $-\overline{uw}$ contours; (c) $-\overline{vw}$ contours; columns, B-L model and two-layer $k - \varepsilon$ model, respectively

nal vortex. At x=0.4 and 0.8, the -uv contours show many similarities between the two results; however, the two-layer model results have larger predicted magnitude near the body surface. This is also true for the $-\overline{uw}$ and $-\overline{vw}$ contours. For x=0.8, it is seen that the two-layer model results generally have smaller extent of contours than the B-L model results. At x=1.1, i. e., in the wake region, quite a few differences are observed between the two results, i. e., the $-\overline{uv}$, $-\overline{uw}$ and $-\overline{vw}$ contours of the B-L model results show much larger magnitude than those of the two-layer model results. As discussed in the workshops^{1),2)}, the B-L model tends to overpredict the Reynolds stresses in the wake region, which results in underestimation of the strength of longitudinal vortex. The present results are consistent with it, i. e., clearer longitudinal vortex is observed in the present two-layer model results than the B-L model results. However further investigation is required for more comprehensive discussion.

Figs. 18 and 19 provide comparison of near-wall velocity profiles between the two-layer and B-L model results at x=0.4 and 0.8, respectively. Both results show characteristic features of the flow in the near-wall region, i.e., the laminar sublayer, the buffer layer, and a part of the fully-turbulent logarithmic layer. As shown in the figures, the present two-layer model approach uses much finer grids near the body surface. At x=0.4, both results leave the logarithmic law at nearly same y^+ for the keel as well as the waterline, where the two-

layer results show closer agreement with the logarithmic line in the region. These aspects generally hold true at x=0.8, where the boundary layer near the waterline is much thicker than that of previous section.

Figs. 20 and 21 provide comparison of solutions on ship hull between the two-layer and B-L model results. In the figures, the contour interval is 0.02 and solid, dash, and dot lines express positive, zero, and negative values, respectively. Surface pressure contours are nearly identical between the two results, and this is consistent with the C_P profiles along the keel and waterline. Generally same aspects are observed in shear stress vectors of the two results; however, the magnitude of vectors are slightly different as shown in the profiles, i. e., the two-layer model results are larger and smaller on the fore and after parts of the body, respectively. Integration of pressure and shear stress distributions over the body surface yields pressure and frictional resistance coefficients, respectively, which are shown in Table. 1 and Fig. 22. The table and figure include the experimental data of Toda et al.¹¹⁾ at Fr =0.1544 and Re = 2,058,700, where CF in the experiment was given by the ITTC 1957 correlation line. Two computations show relatively good agreement with the experimental data; however, the two-layer model gives somewhat smaller frictional resistance than the B-L model, which results in closer agreement of the present results with the experimental data.



Fig. 18 Near-wall velocity profiles at x=0.4: (a) keel; (b) waterline



Fig. 19 Near-wall velocity profiles at x=0.8: (a) keel; (b) waterline

An Application of Two-Layer $k - \varepsilon$ Model to Ship Flow Computation





0.9784

0.9916

5.0888

4.9832

4.1104

3.9916

B-L Model

2-L Model

Fig. 22 Comparison of resistance coefficients

-121 -

7. Concluding remarks

This work presents a numerical method for calculating ship boundary layers and wakes for zero Froude number. The Reynolds-averaged Navier-Stokes and continuity equations are solved with the two-layer $k-\epsilon$ turbulence model. The results are validated through comparisons with the experimental data for the Series 60 C_B =.6. ship model and results of the precursory wall function as well as the Baldwin-Lomax model approaches. Three turbulence models yield satisfactory results; however, the present and Baldwin-Lomax model results indicate improved resolution of the flow close to the hull and wake centerplane due to near-wall turbulence modeling.

Although three approaches satisfactory predict the general feature of the flow, detailed resolution is still lacking. Based on the studies^{1),2)}, more effort must be focused on the turbulence modeling. For instance, the Reynolds stress model definitely show superior performance, and warrant further investigation including more simplified algebraic models or nonlinear $k-\varepsilon$ model. Also more detailed evaluation of the present method is necessary in practical and complicated flows. Development of an extended version for the free-surface flow around a yawed ship is currently in progress.

8. Acknowledgments

The author wishes to thank Prof. Y. Himeno at University of Osaka Prefecture for his valuable discussions and encouragement. The computations were performed on the Osaka University Computer Center and the National Center for Supercomputing Applications.

References

- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.
- 2) Proc. Workshop on Wave Resistance and Viscous Flow, Tokyo, July 1994.
- 3) Tahara Y. and Stern, F., "A Large-Domain

Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.

- 4) Patel, V. C., Chen, H. C. and Ju, S., "Ship Stern and Wake Flows: Solutions of the Fully-Elliptic Reynolds-Averaged Navier-Stokes Equations and Comparisons with Experiments," J. Computational Physics, Vol. 88, No. 2, June 1990, pp. 305-336.
- Tahara, Y., Stern, F. and Rosen, B., "An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," J. Computational Physics, Vol. 98, No. 1, January 1992, pp. 33-53.
- 6) Tahara Y. and Stern, F., "Validation of an Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," to appear J. Computers and Fluids.
- Tahara, Y., "Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments," J. Kansai Society of Naval Architects, Japan, No. 220, September 1993, pp. 29-47.
- Chen, H. C. and Patel, V. C., "The Flow Around Wing-Body Junctions," Proc. 4th Symp. Num. Phys. Aspects Aerodyn. Flows, Cebeci ed., Long Beach, CA, 1989.
- 9) Kim, S., "Numerical Studies of Three-Dimensional Flow Separation," Ph. D. Thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- Kim, W. J., "An Experimental and Computational Study of Longitudinal Vortices in Turbulent Boundary Layers," Ph. D. Thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- 11) Toda, Y., Stern, F. and Longo, J., "Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B =.6 Ship Model-Part 1: Froude Numbers .16 and .316," J. Ship Research, Vol. 37, No. 4, December 1992, pp. 360-377.
- 12) Toda, Y., Stern, F., Tanaka, I., and Patel, V. C., "Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B=.6 Model Ship With and Without Propeller," J. Ship Research, Vol. 34, No. 4, 1990, pp. 225-252.

-122 -

関西造船協会誌 第224号 平成7年9月

Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht - For a Canoe Body Case - *1

Yusuke TAHARA (Member) *2

A numerical method is developed for calculating boundary-layer and wake flows around the International America's Cup Class (IACC) sailing yacht for zero Froude number. The Reynolds-averaged Navier-Stokes and continuity equations are solved with the Baldwin-Lomax turbulence model, using a body conforming grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. The computational domain includes both port and starboard sides of the hull in order to account for the ship flow with yaw angle. An overview is given for the present approach, and numerical results are presented and discussed for the canoe body of IACC sailing yacht for zero and nonzero yaw angles. Lastly, some concluding remarks are made, including the requirements and prognosis for extension of the present method for the complete hull geometry with keel and rudder.

Keywords : Boundary-layer Flow, Navier-Stokes Equations, America's Cup Sailing Yacht, Finite-analytic Scheme

1. Introduction

The America's cup can be viewed as a colossal hightech contest where the best technical talents and resources of each competing country are applied to a design problem which is rigidly constrained by rules. Fast boat must have efficient sail and low drag forces, as well as stability and maneuverability. One of the most time-consuming problem designers face is controlling the drag forces. Since development of the Stars and Stripes '87, more and more computational methods have been used in the boat design in conjunction with the experimental approaches $^{1)2}$. Designers have demanded for the numerical tools which accurately estimate the hydrodynamic forces acting on the hull. The present study is central to the aforementioned problem; i.e., it concerns the development of a numerical approach for calculating boundary-layer and wake flows around the International America's Cup Class (IACC) sailing yacht.

*2 University of Osaka Prefecture

Hydrodynamics associated with a sailing yacht is very complicated. The sailing yacht heels in response to the heeling moment of sails, and develop sideforce to balance sideforce of the sails. The appendages like keel and rudder of the IACC sailing yacht also produce complicated flow field. Milgram and Frimm³⁾ decomposed the total resistance acting on the hull into five parts, i.e., the frictional drag of the hull, the upright residuary resistance of the entire vessel, the frictional and interference drag of appendages, the resistance due to heel and side force production (induced drag), and the resistance due to sea waves (added resistance). It is clear that the way to determine those terms is not unique, because each of the components is coupled. Determination of the resistance components and their interaction is one of the most important problems.

Initially, experimental approach had been used to study the problem associated with the sailing yacht, mainly focusing on the scale effects of the model testing. More recently, quite a few computational approaches have been developed not only for engineering prediction of overall performance characteristics, but for research study to investigate the underlying flow physics (see Proc.²⁾ for recent examples). Most of

^{*1} Read at the Spring Meeting of Kansai Society of Naval Architects, Japan, May 26, 1995, Received June 7, 1995

the computational studies reported so far for sailing yachts are based on the inviscid-flow methods. For instance, Xia and Larsson⁴⁾ and Rosen et al.⁵⁾⁶⁾ developed the Dawson-type panel methods including freesurface effects and lifting effects. Nakatake et al.⁷⁾ and Tahara⁸⁾ also developed the similar approaches, although the results had been presented only for the Wigley hull. The panel method is currently the most popular application of the inviscid-flow methods, because of its flexibility to the geometrical complexities. However, the results show clear difficulties in predicting detailed aspects of the flow, which are due to the lack of viscous effects in the theory.

On the other hand, considerable efforts have been put forth in the development of viscous-flow methods for ship boundary layers and wakes. Initially, threedimensional integral and differential boundary-layer equation methods were developed (e.g., see Ikehata and Tahara⁹) or Tanaka et al.¹⁰) for relatively recent examples); however, these were found to be inapplicable near the stern and in the wake. More recently, efforts have been directed towards the development of Navier-Stokes (NS) and Reynolds-averaged Navier-Stokes (RaNS) equation methods¹¹⁾¹²⁾. For instance, Tahara et al.¹³)¹⁷ developed both interactive and large-domain approaches for calculating ship boundary layers and wakes for zero and nonzero Froude number (Fn). In the studies $^{11}^{-17}$, comparisons with the experimental data indicate that many important features of the flow are adequately simulated, although more effort must be focused on the turbulence model (see Tahara¹⁷) for more related issues).

In this study, a numerical method is developed, which is based on the extension of Tahara¹⁵), for calculating boundary-layer and wake flows around the IACC sailing yacht for zero Fn, i.e., wave effects are not included in the theory and free surface is treated as a symmetric boundary. The RaNS and continuity equations are solved with the Baldwin-Lomax turbulence model, using a body conforming grid, finiteanalytic discretization, and a PISO-type velocitypressure coupling algorithm. The computational domain includes both port and starboard sides of the hull in order to account for the ship flow with yaw angle. In the following, the computational method, conditions, and grid are described and numerical results are presented and discussed for the canoe body of the IACC sailing yacht for zero and nonzero yaw angles. Lastly, some concluding remarks are made, including the requirements and prognosis for extension of the present method for the complete hull geometry with keel and rudder.

2. Computational Method

The unsteady RaNS and continuity equations with the Baldwin-Lomax turbulence model are solved for the mean-velocity components (U,V,W), pressure p, and eddy viscosity ν_t . The transport equations for (U,V,W) are written in Cartesian coordinates (X,Y,Z) in the physical domain and partially transformed into numerically generated boundary-fitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) . Variables are nondimensionalized using the freestream velocity U₀ (=1), body length L (=1), and density ρ . The transformed equations are solved using a regular grid, finite-analytic discretization, a PISO-type velocitypressure coupling algorithm, and the method of lines. Although the present solutions are for steady flow, the equations are solved in unsteady form with time serving as a convergence parameter. For more detailed equations, solution procedure and references, see Tahara et al. $^{15)16}$.



Fig. 1 Definition sketch of coordinate system, solution domain and boundaries

Consider a ship fixed in the uniform onset flow $U_0 = (U_0 \cos \alpha, U_0 \sin \alpha, 0)$ as depicted in Fig.1 (where α is the angle of attack). In terms of the notation of Fig.1, the boundary conditions on each of the boundaries are described as follows: on inlet plane Si

and outer boundary So, the boundary conditions are provided on the basis of the freestream values, i.e., $U=U_0 \cos \alpha$, $V=U_0 \sin \alpha$, W=0 and p is linearly extrapolated from the interior nodes; on exit plane Se, axial diffusion is negligible so that the exit conditions used are $\partial^2(U,V,W)/\partial x^2 = \partial p/\partial x=0$; on waterplane Sw, the conditions imposed are $\partial(U,V,p)/\partial z=W=0$; and on body surface Sb, boundary conditions used are $U=V=W=\partial p/\partial n=0$ (where n is normal to the body). In this study, the initial conditions are taken from the freestream values.

3. Results and discussion

In the present study, the computational results are presented for the canoe body named MX01, which is one of the hull forms selected by the PACT (Partnership for America's Cup Technology), founded for the US. America's Cup Defense in February 1990. Because the present study is not allowed to disclose details of the hull information as well as the experimental data, the discussions of the computational results to follow will mainly be focused on the flow physics associated with the present problem.

The computational grids were obtained using the elliptic-algebraic method¹⁶). The inlet, exit and outer boundaries are located at x=(-0.4, 2.0) and r=1, respectively. The first grid point off the body surface is located in the viscous sublayer, i.e., $y^+ < 2$. The numbers of grids used are 90x30x29 (=78,300) in the axial, radial, and circumferential directions, respectively. Fig.2 shows partial views of the grid for the body surface and typical body and wake crossplanes. The conditions for the computation were as follows: L=1; U₀=1; Rn=4,500,000. The supercomputer (SX-3R) time and central memory were about 1 hour and 40 MB. The convergence criterion was that the change in solution be less than about 0.05% for all variables, which was satisfied in 1000 global iterations.

Figs.3 through 5 show the axial-velocity (U) contours, crossplane-velocity (V-W) vectors, pressure (Cp=p/.5) contours, and axial-vorticity (ω_x) contours for several representative sections. The surfacepressure distributions, the limiting streamlines, the wall-shear-stress (C $_{\tau} = \tau_{\omega}/.5\rho U_0^2$) contours, and resistance components are compared between zero and nonzero yaw angles in Figs.6 and 7.



Fig. 2 Computational grid: (a) body surface; (b) body crossplane(X=0.5); and (c) wake crossplane(X=1.1)

3.1 Zero yaw angle $(\alpha=0^\circ)$

At x=0, the results show that the flow is dominated by stagnation effects associated with the close proximity of the bow. The U contours are less that unity through the extent of the present region. The Cp contours are similar to those for U, but with reversed trend in magnitude, and display quite large values in the inner part. The V-W vectors show the double model stagnation flows, i.e., the outward flows due to displacement effects of the hull.

At x=0.25, the U contours display the thickening

of the boundary layer, especially near the keel, which is due to the flow convergence towards the keel. The Cp contours show the lowest value near the keel, and the highest value near the waterline. All V-W vectors show the downward flow. No bow bilge vortex is observed unlike for the case of the commercial ship form, e.g., the Series 60 or HSVA tanker form (see Tahara et al.¹³⁾⁻¹⁷⁾ for more details and references).

At x=0.5, the U contours show continuous growth of the boundary layer and the convergence of the flow near the keel. The boundary layer is quite thick near the keel, and thin near the waterline. The V-W vectors display decreased magnitude and the Cp contours decreasing values from x=0.25 to the present section. The V-W vectors are all downward. The ω_x contours are anti-symmetric with respect to the centerplane, and the negative and positive values are observed near the waterline, and the positive and negative values near the midgirth, for the port and starboard sides, respectively.

At x=0.75, the U contours display the thick boundary-layer characteristics. The boundary layer is thicker near the waterline than the keel, which is due to the convergence of the flow near the waterline. All V-W vectors are directed upward, and the Cp contours show negative values everywhere, and increasing values from x=0.5 to the present section, where the lowest value is observed near the keel. The ω_x contours show relatively plane distributions as compared to those of the commercial ship form, where the stern bilge vortex is usually observed.

At x=1 and 1.1, the U and Cp contours display merging of the boundary layer into the wake and its initial evolution. The Cp contours are all positive through the extent of the present region, and show the highest value near the stern for x=1, and the wake centerplane near the free surface for x=1.1. The V-W vectors are all upward. As mentioned for the previous section, the V-W vectors and the ω_x contours do not display clear longitudinal vortex unlike those of the commercial ship form.

The surface Cp contours (Fig.6a, $\alpha=0^{\circ}$) show the lowest value near the largest draft of the hull. The limiting streamlines (Fig.6b, $\alpha=0^{\circ}$) show the convergence of the flow towards the waterline in the afterbody of the hull, which is correlated with the thicker boundary layer displayed in the U contours at x=0.75. The limiting streamlines do not indicate the obvious flow separation. The C_{τ} contours (Fig.6c, $\alpha=0^{\circ}$) show the highest value near the waterline in the forebody of the hull, and decreasing values towards the stern, which are related to the development of the boundary layer. **3.2** Nonzero yaw angle ($\alpha=10^{\circ}$)

At x=0, the results for α =10° display the similar stagnation effects described for α =0°, but with somewhat leftward shift. The U contours are less that unity through the extent of the present region, and the Cp contours are similar to those for U with reversed trend in magnitude, which also display quite large values in the inner part. The V-W vectors are all rightward and completely different from those for α =0°.

At x=0.25, quite a few differences of the flow field are observed between $\alpha = 0^{\circ}$ and $\alpha = 10^{\circ}$ regarding the U and Cp contours as well as the V-W vectors. The U and Cp contours for $\alpha = 10^{\circ}$ near the port-side waterline display the similar stagnation effects of the previous section, where the U contours are less than unity through the extent of the region, and the Cp contours show quite large values near the port-side waterline. The Cp contours for $\alpha = 10^{\circ}$ show more complexities than those for $\alpha=0^{\circ}$, i.e., the values near the body are mostly positive at the port side except near the keel, and negative at the starboard side. The V-W vectors are leftward on the free surface near the portside waterline, and suddenly directed rightward at larger depths. The V-W vectors at the starboard side are all rightward, and slightly directed towards the starboard- side free surface. The U contours also show that the boundary layer is somewhat thicker at the starboard side, which is due to the above-mentioned flow convergence.

At x=0.5, the U contours for α =10° show continuous growth of the boundary layer and the convergence of the flow near the starboard-side waterline. The boundary layer is thin at the port side, and thick near the starboard-side waterline. The V-W vectors for α =10° are all rightward, and downward and upward near the port- and starboard-side waterlines, respectively, which results in the convergence of the flow near the starboard-side waterline. The Cp contours for α =10° show decreasing values from x=0.25 to the present section. The ω_x contours for α =10° are quite different from those for α =0°, i.e., the values are all positive in the present region, which is related to the generation of the anti-clockwiseaxial vortex.

At x=0.75, the U contours for $\alpha = 10^{\circ}$ display the



Fig. 3 Comparison of solutions for $\alpha=0^{\circ}$: (a) axial-velocity contours; (b) crossplane-velocity vectors; and (c) Cp contours; rows, X=0(FP), X=0.25, X=0.50, X=0.75, X=1.0(AP), and X=1.1, respectively



Fig. 4 Comparison of solutions for $\alpha=10^{\circ}$: (a) axial-velocity contours; (b) crossplane-velocity vectors; and (c) Cp contours; rows, X=0(FP), X=0.25, X=0.50, X=0.75, X=1.0(AP), and X=1.1, respectively

-128-



Fig. 5 Comparison of axial-vorticity contours (ω_x): (a) $\alpha=0^{\circ}$; and (b) $\alpha=10^{\circ}$; columns, X=0.50, X=0.75, and X=1.1, respectively

thick boundary layer near the starboard-side waterline, and thin boundary layer near the port-side waterline, which are due to the convergence of the flow near the starboard-side waterline as shown in the V-W vectors. The V-W vectors for $\alpha=10^{\circ}$ are mostly rightward and upward, but directed leftward near the starboard waterline. The Cp contours for $\alpha=10^{\circ}$ mostly show negative values except near the starboard-side waterline, where the values are generally increasing from x=0.5 to the present section. The ω_x contours for $\alpha=10^{\circ}$ are mostly positive, but negative values are also observed in the region very close to the hull at the starboard side. The ω_x contours show larger extent at the starboard side than the port side.

At x=1 and 1.1, the U and Cp contours for $\alpha=10^{\circ}$ display the merging of the boundary layer into the wake and its initial evolution, which are similar to those for $\alpha=0^{\circ}$ but with different shape of the extents. The U contours for $\alpha=10^{\circ}$ show the slower recovery at the starboard side than the port side. The Cp contours for $\alpha=10^{\circ}$ display larger extents at the starboard side than the port side, i.e., the values are higher at the starboard side than the port side with respect to the centerplane. The Cp contours for $\alpha=10^{\circ}$ have increasing values at x=1 and decreasing values at x=1.1 from the previous sections. The V-W vectors for $\alpha=10^{\circ}$ are all rightward and upward, which is correlated with the general aspects of the boundary layer and wake displayed in the U contours. Like the results for $\alpha=0^{\circ}$, the V-W vectors for $\alpha=10^{\circ}$ do not show clear longitudinal vortex; however, the ω_x contours for $\alpha=10^{\circ}$ display the region of positive values, which indicates the existence of weak anti-clockwise longitudinal vortex in the wake.

The surface Cp contours (Fig.6a, $\alpha = 10^{\circ}$) show completely different aspects from those for $\alpha = 0^{\circ}$. In the forebody of the hull, the Cp contours have higher values at the port side than the starboard side with respect to the centerline; and the reverse holds true in the afterbody of the hull. The lowest value of the Cp contours are observed near the largest draft of the hull, which is in the similar location as that for $\alpha=0^{\circ}$ but slightly shifted towards the port side. The limiting streamlines (Fig.6b, $\alpha=10^{\circ}$) show the conver-



Fig. 6 Comparison of solutions on body surface: (a) Cp contours; (b) limiting streamlines; and (c) C_{τ} contours; columns, $\alpha=0^{\circ}$; and $\alpha=10^{\circ}$, respectively

gence of the flow towards the starboard-side waterline in the afterbody of the hull, which is correlated with the thicker boundary layer in the region displayed in the U contours at x=0.75. The limiting streamlines for $\alpha = 10^{\circ}$ do not indicate the obvious flow separation but show more complex aspects than those for $\alpha = 0^{\circ}$. The C_{τ} contours (Fig.6c, $\alpha=10^{\circ}$) also display quite different aspects from those for $\alpha=0^{\circ}$, i.e., the contours show the highest value near the port-side waterline in the forebody of the hull, and decreasing values towards the starboard-side waterline in the afterbody of the hull, which are related to the development of the boundary layer. As is the case for the Cp, the C_{τ} distributions on the hull also produce the lifting and drag forces, which is in more complicated manner than that related to the Cp, because the direction of the wall-shear-stress vectors must be considered in the analysis.

3.3 Lifting and drag forces

Integration of the pressure and shear stress distributions of the present computational results over the body surface yields the pressure and frictional resistance coefficients, respectively, which are shown in Fig. 7. It is seen that the frictional part of Cd (denoted as CF in the figure) is almost equal as yaw angle α increases, whereas the pressure part (denoted as CP) indicates nearly parabolic increase. On the other hand, the frictional part of Cl is negative and its magnitude linearly increases as α increases, whereas the pressure part is always positive and linearly increases. The lifting line theory gives an estimation of Cl and Cd for the wings of finite span, such that Cl is proportional to α and Cdi, i.e., the induced drag, is proportional to Cl^2 . In the present results, the increase of CP in Cd can be correlated with the induced drag, which is due to the generation of the longitudinal vortex. The above will provide some physical understanding about the present results; however of course, the drag components in the present problem include more complex aspects, i.e., the pressure part in Cl or Cd is influenced by the boundary-layer flow, which is not considered in the inviscid-flow theory. Here, it must be noted that the present computation gives relatively



Fig. 7 Comparison of resistance coefficients

large frictional drag as compared to the Schoenherr's value. This may be due to the present grid resolution or turbulence model, which will be issues of the present study for future improvement.

4. Concluding remarks

This work presents a numerical method for calculating boundary-layer and wake flows around the IACC sailing yacht for zero Fn. The RaNS and continuity equations are solved with the Baldwin-Lomax turbulence model, using a body conforming grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. The computational domain includes both port and starboard sides of the hull in order to account for the ship flow with yaw angle. The numerical results presented for the canoe body for zero and nonzero yaw angles are found to be very encouraging. However, complete validation of the present method was not possible, due to the limited experimental data. Currently development of an extended version of the present approach for the complete hull geometry with keel and rudder is in progress. This extension will enable more detailed validation of the present method and an evaluation of interference drags among the hull and appendages.

With regard to the capability of the present method for nonzero yaw angle, a complete evaluation requires a detailed comparative study, which explicates the complicated underlying flow physics associated with the practical ship geometry, such as the unsteady characteristics of the flow field or the three-dimensional flow separation and vortex shedding. This statement is supported by the recent experimental and computational studies. For instance, Nishio¹⁸ investigated

the three-dimensionally separated flow around a prolate spheroid and a ship model at incidence, where the detailed measurements were made on the mean velocity and pressure fields as well as the surface pressure distributions. Ohmori and Miyata¹⁹⁾ developed a numerical method for the oblique tow simulation, and the results presented for three ship models show satisfactory agreement with the experimental data regarding the lateral force and yawing moment. The work of Fuwa²⁰⁾ and Matsumura et al.²¹⁾ are precursory to those studies. In the author's research group, an extended version of the large-domain approach of Tahara and Stern¹⁶⁾ for zero and nonzero Fn for a yawed ship is currently under development, which will be reported in a future publication. Also, recent progress on the two-layer k- ε turbulence model approach by Tahara¹⁷) offers encouragement in extending the techniques for the present application.

The wave/boundary-layer and wake interaction is also a very important factor to estimate the resistance components and their interaction. This statement is supported by the recent work of the interactive and large-domain approaches by Tahara et al.¹³⁾¹⁴⁾¹⁶. Based on the studies, the wave/boundary-layer and wake interaction is most significant at the bow, and on the afterbody of the ship hull and in the wake. In these areas, both the viscous and inviscid flows are altered due to their interactions. In the latter case, most of the interaction can be explicated as a result of the wave elevations, the wave-induced pressure gradients, and the displacement effects of the boundary layer. Furthermore, the work of Tahara⁸, which concerns a development of a Dawson-type panel method, implies that the above-mentioned flow aspects will be more complicated for the case with nonzero yaw angle.

Finally, the issues that need to be addresses while further developing and validating the present approach are summarized as follows: further validation of the present numerical method through the comparisons with available experimental data; further evaluation of the present method in conjunction with the extension for the complete hull with keel and rudder; inclusion of free-surface effects; and modification of the computational grid and turbulence model. Also it is of interest to introduce the advanced convergenceacceleration technique regarding the overall solution procedure, especially for the velocity-pressure coupling.

5. Acknowledgments

The author wishes to thank Prof. Y. Himeno at University of Osaka Prefecture for his valuable discussions and encouragement. The computations were performed on the Osaka University Computer Center.

References

- "Collected Papers on Technology for Stars & Stripes", Science Applications International Corporation, Annapolis, MD, January 1988.
- 2) Proc. The Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, MD, January 1993.
- Milgram, J.H. and Frimm, F.C., "Elements of Resistance of IACC Yachts," Proc. The Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, MD, January 1993, pp. 223-229.
- Xia, F. and Larsson, L., "A Calculation Method for the Lifting Potential Flow Around Yawed, Surface-Piercing 3-D Bodies," Proc. Sixteenth Symposium on Naval Hydrodynamics, Berkeley, 1986, pp. 583-597.
- 5) Boppe, C.W., Rosen, B.S., Laiosa, J.P. and Chance, B., Jr., "Stars & Stripes '87: Computational Flow Simulations for Hydrodynamic Design," The Eighth Chesapeake Sailing Yacht Symposium, Annapolis, MD., 1987.
- 6) Rosen, B.S., Laiosa, J.P., Davis, W.H. and Stavetski, D., "SPLASH Free-Surface Flow Code Methodology for Hydrodynamic Design and Analysis of IACC Yachts," Proc. The Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, MD, January 1993, pp. 35- 49.

- Nakatake, K., Komura, A., Ando, J. and Kataoka, K., "On the Flow Field and Hydrodynamic Forces of an Obliquing Ship," Transaction of the West-Japan Society of Naval Architects, No. 80, 1990, pp. 1-12 [Japanese].
- 8) Tahara, Y., "A Boundary-Element Method for Calculating Free-Surface Flows around a Yawed Ship," J. Kansai Society of Naval Architects, Vol. 218, 1992, pp. 55-67.
- Ikehata, M. and Tahara, Y., "Influence of Boundary Layer and Wake on Free Surface Flow around a Ship Model," J. Society of Naval Architects of Japan, Vol. 161, 1987, pp. 49-57 [Japanese]; also Naval Architecture and Ocean Engineering, Vol. 26, 1988, pp. 71-78 [English].
- 10) Tanaka, H., Toda, Y. and Suzuki, T., "A Simple Integral Method for Thick Boundary Layer for Predicting Axial Velocity Distribution around Ship Stern," J. Kansai Society of Naval Architects, No. 221, 1994, pp. 17-24 [Japanese].
- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.
- Proc. Workshop on Wave Resistance and Viscous Flow, Tokyo, July 1994.
- Tahara, Y., Stern, F. and Rosen, B., "An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," J. Computational Physics, Vol. 98, No. 1, January 1992, pp. 33-53.
- 14) Tahara, Y. and Stern, F., "Validation of an Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," J. Computers and Fluids, Vol. 23, No. 6, 1994, pp. 785-816.
- 15) Tahara, Y., "Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments," J. Kansai Society of Naval Architects, No. 220, 1993, pp. 29-47.
- 16) Tahara, Y. and Stern, F., "A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994, pp. 45-55.
- 17) Tahara, Y., "An Application of Two-Layer k-ε Model to Ship Flow Computation," to appear J. Society of Naval Architects of Japan, 1995. 18.
- 18) Nishio, S., "Study on the Three Dimensional Separated Flow around Prolate Spheroid and Ships

at Incidence," Doctoral thesis, Osaka University, Osaka, 1990 [Japanese].

- 19) Ohmori, T. and Miyata, H., "Oblique Tow Simulation by a Finite-Volume Method," J. Society of Naval Architects of Japan, Vol. 173, 1993, pp. 27-34.
- 20) Fuwa, T., "Hydrodynamic Forces Acting on a Ship in Oblique Towing," J. Society of Naval Architects of Japan, Vol. 134, 1973, pp. 135-147 [Japanese].
- 21) Matsumura, K., Tanaka, I., Oki, T. and Kishi, S., "On the Nonlinear Lift Characteristics of Slender Bodies at Incidence," J. Society of Naval Architects of Japan, Vol. 154, 1983, pp. 283-293 [Japanese].

討 論

[討論] (日立造船(株)技術研究所)村上光功 最新の理論に基づき詳細な計算結果をお示しになった

ことに敬意を表します。以下の点につきご教示下さい。 1) 今回は第一段階として、Appadape なしの船体周 りの流端について示されました。しかし、実際の場合 には、keel 他 sepanation を積極的に誘起するものも あり、更に波中での3次元運動も十分考慮できるとこ ろであります。この様な問題に対して、実用的観点か らも、どのように今後進めていかれるのでしょうか、 ご見解をご教示賜わりたく存じます。

2) 近年、船尾周りの複雑な流れ、とくに船尾縦渦の 問題が盛んに論じられております。著者の方法(理論) によりこの方面の問題についてはどの様に議論される のでしょうか。

[回答]

(1)

アペンテージ(キールとラダー)を装着した場合の計 算は実際に現在進行中です。計算労力が増大するのが 難点ですが、その詳細な評価も含め近い将来ご紹介す る予定です。本研究のプロジェクトの目的はそのレベ ルまで到達することであり、今回ご紹介した研究はそ の第1ステップであるとお考え下さい。

また3次元運動の考慮ですが、著者が用いている解 法の基礎方程式は全て非定常問題として定式化されて おり、よって Time Accurate な解法へ拡張することは 原理的に難しいとは思われません。しかしながら、そ の場合論点となるのは計算労力の増加および結果の詳 細な評価です。利用可能な実験値も限られており、総 合的な研究プロジェクトを開始するためには周到な準 備が必要であると思われますが、近い将来ぜひ挑戦し てみたいと考えています。 (2)

著者等の研究グループでは、船尾流場における計算 精度向上の鍵は乱流モデルにあるという見解をもって います。今回ご紹介した計算法は、基本計算スキームは そのままに各種乱流モデルを独立搭載することが可能 であり、その実例は田原による最近の計算(日本造船学 会論文集第177号掲載予定)でもご紹介しました。近 い将来、船尾縦渦の表現力が争点となるタンカー船型 を対象とした研究をご紹介する予定です。そこでは等 方性及び非等方性乱流モデルの性能評価を行ないます。

Applications of Isotropic and Anisotropic Turbulence Models to Ship Flow Computation *1

Yusuke TAHARA (Member) *2 and Yoji HIMENO (Member)*2

This paper presents applications of isotropic and anisotropic turbulence models in computation of ship boundary-layer and wake flows. The fully-elliptic Reynolds-averaged Navier-Stokes and continuity equations are solved with several kinds of turbulence models, using a regular grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. An overview is given of the present numerical method, and results are presented and discussed for the SR196a tanker form, including detailed comparisons with available experimental data. Lastly, some concluding remarks are made concerning limitations, requirements and prognosis for improvements of the present turbulence models.

Keywords : Ship Boundary Layers, Anisotropic Turbulence Model, Two-layer k- ε Model, Tanker Form, Finite-Analytic Scheme

1. Introduction

Ship flows near the stern must accurately be predicted in order to estimate the propeller performance. The conventional tanker form is characterized by its complicated hull shape, which results in considerably thick boundary layer and strong stern bilge vortex, a longitudinal vortex in near wake. The experiments observe quite interesting aspects of the wake flow, such as the "hook shape" of the axial-velocity contours. Many numerical schemes have been developed to reproduce the complicated flow field, but most of them turned out to be failure due to inadequate accuracy of the numerical scheme, grid resolution, and turbulence modeling. The recent workshops¹⁾²⁾ stated that turbulence modeling may be the most important issue for the problem. Quite a few researchers¹⁾²⁾³⁾ have focused on modification of the Baldwin-Lomax model to provide appropriate eddy-viscosity distribution, which is able to reproduce the "hook shape." Although satisfactory modifications are still under development, the approaches attract many ship designers, who have demanded for handy turbulence models in practical purpose.

On the other hand, the recent studies have shown that the conventional eddy-viscosity assumption indicates many difficulties to describe the flow with strong secondary motion including the above-mentioned ship flows. This implies the need for higher-order turbulence models, i.e., the anisotropic models. Several studies have been reported for the second-order Reynolds-stress closure models, including a few for the application to ship flow¹⁾. The results presented for the tanker forms were shown to be very promising; however, inevitable difficulties were also pointed out, i.e., the lower computational stability and efficiency. In practical use of the anisotropic turbulence model, it is of great interest to develop a simplified form with careful consideration and evaluation of its performance and limitation. This may be an important approach for the present and future work as long as the practical purpose is concerned, while the nonzeroequation turbulence model must also be investigated.

This paper presents applications of isotropic and anisotropic turbulence models in computation of ship boundary-layer and wake flows. The fully-elliptic RaNS and continuity equations are solved with several kinds of turbulence models, i.e., the Baldwin-Lomax model, the two-layer k- ε model used by Tahara⁴, and a modified Baldwin-Lomax model proposed by Himeno³. Numerical results are obtained using

^{*1} Read at the Joint Autumn Meeting of Three Societies of Naval Architects in Japan, November 16, 1995, Received December 1, 1995

^{*2} Osaka Prefecture University
a regular grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. An overview is given of the present numerical method, and results are presented and discussed for the SR196a tanker form for zero Froude number, i.e., the free surface is considered to be flat and treated as a plane of symmetry, including detailed comparisons with available experimental data. Lastly, some concluding remarks are made concerning limitations, requirements and prognosis for improvements of the present turbulence models.

2. Governing equation

The non-dimensional RaNS equations for unsteady, three-dimensional incompressible flow can be written in Cartesian tensor notation as

$$\sum_{i=1}^{3} \frac{\partial U_i}{\partial x^i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \sum_{j=1}^3 \left(U_j \frac{\partial U_i}{\partial x^j} + \frac{\partial \overline{u_i u_j}}{\partial x^j} \right) + \frac{\partial p}{\partial x^i} - \frac{1}{\mathrm{Rn}} \nabla^2 U_i = 0$$

with $\nabla^2 = \sum_{j=1}^3 \frac{\partial^2}{\partial x^j \partial x^j}$ (2)

where $U_i = (U, V, W)$ and $u_i = (u, v, w)$ are the Cartesian components of mean and fluctuating velocities, respectively, normalized by the reference velocity U_0 , $x_i = (X, Y, Z)$ are the dimensionless coordinates normalized by a characteristic length L, p is the pressure normalized by ρU_0^2 , $Rn = U_0 L/\nu$ is the Reynolds number, ν is the kinematic viscosity, and the barred quantities $\overline{u_i u_j}$ are the Reynolds stresses normalized by U_0^2 . Equations (1) and (2) can be solved for U_i and p when a suitable turbulence model is employed to provide the Reynolds-stress components. **2.1** k- ε model

If the Reynolds stresses $\overline{u_i u_j}$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity ν_t , i.e.

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i} \right) - \frac{2}{3} \delta_{ij} k \tag{3}$$

where $k=(\overline{uu}+\overline{vv}+\overline{ww})/2$ is the turbulent kinetic energy, the eddy-viscosity distribution is necessary information to effect the closure. In the k- ε turbulence

model, eddy viscosity is defined by $\nu_t = C_{\mu} k^2 / \varepsilon$, where ε is the rate of turbulent energy dissipation. k and ε are obtained from the transport equations

$$\frac{\partial k}{\partial t} + \sum_{j=1}^{3} \left(U_j - \frac{1}{\sigma_k} \frac{\partial \nu_t}{\partial x^j} \right) \frac{\partial k}{\partial x^j} - \frac{1}{R_k} \nabla^2 k - G + \varepsilon = 0$$
(4)

$$\frac{\partial \varepsilon}{\partial t} + \sum_{j=1}^{3} \left(U_j - \frac{1}{\sigma_{\varepsilon}} \frac{\partial \nu_t}{\partial x^j} \right) \frac{\partial \varepsilon}{\partial x^j} - \frac{1}{R_{\varepsilon}} \nabla^2 \varepsilon - C_{\varepsilon 1} \frac{\varepsilon}{k} G + C_{\varepsilon 2} \frac{\varepsilon^2}{k} = 0$$
(5)

where R_k and R_{ε} are the effective Reynolds numbers defined by $1/R_{\phi}=1/Rn + \nu_t/\sigma_{\phi}$, $\phi=(k,\varepsilon)$, and

$$G = \frac{1}{2}\nu_t \sum_{i=1}^{3} \sum_{j=1}^{3} \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i}\right)^2 \tag{6}$$

is the rate of production of k, and $(C_{\mu}, C_{\epsilon 1}, C_{\epsilon 2}, \sigma_k, \sigma_{\epsilon})$ are constants whose values are (0.09, 1.44, 1.92, 1.01, 1.3). As is well-known, we have difficulties in use of the equations near the solid boundaries, since viscous effects very close to the wall are not considered in the model. The two-layer approach has been proposed in order to solve this problem⁵⁾⁶⁾⁷⁾.

In the two-layer approach used in the present study, different equations are solved in the outer and near-wall layers. In the outer layer, eddy viscosity is defined by $\nu_t = C_{\mu} k^2 / \varepsilon$, where k and ε are obtained from equations (4) and (5). In the near-wall layer, only k is obtained from the transport equation (4), then ν_t and ε are given by $\nu_t = C_\mu \sqrt{k} l_\mu$ and $\varepsilon = k^{3/2}/l_\varepsilon$, where l_μ and l_ε are the length scales defined by $l_{\mu}=C_l y [1-exp(-R_y/A_{\mu})]$ and $l_{\varepsilon} = C_l y [1 - exp(-R_y/A_{\varepsilon})]$. $R_y = Rn\sqrt{ky}$ is the turbulent Reynolds number and y is the distance from the wall. $C_l = \kappa C_{\mu}^{-3/4}$, $A_{\mu} = 70$, and $A_{\varepsilon} = 2C_l$ are constants used by Chen and Patel⁶⁾, where $\kappa = 0.418$ is the von Karman constant. The value of C_l was determined to ensure a smooth eddy-viscosity distribution at the boundary of the near-wall and outer layers. A_{μ} was determined from numerical tests so as to recover the additive constant B=5.45 in the logarithmic law in the case of a flat plate boundary layer. The value of A_{ε} was assigned so as to recover the proper asymptotic behavior $\epsilon = 2\nu k/y^2$ in the sublayer. Note that

the near-wall layer includes the viscous sublayer, the buffer layer, and a part of the fully-turbulent logarithmic layer.

2.2 Baldwin-Lomax model

The Baldwin-Lomax model is an algebraic scheme that makes use of a two-layer isotropic eddy-viscosity formulation. In this model, the eddy viscosity is evaluated as follows:

$$\nu_t = \begin{cases} (\nu_t)_{inner} & y \le y_c \\ (\nu_t)_{outer} & y > y_c \end{cases}$$
(7)

where y is the distance normal to the wall surface and y_c is the minimum value of y where both the inner and outer viscosities match. The inner viscosity follows the Prandtl-Van Driest formula, i.e., $(\nu_t)_{inner} = l^2 |\omega|$, where $l = \kappa y [1 - exp(-y^+/A^+)]$ is the turbulent length scale for the inner region, κ and A^+ are model constants, $|\omega|$ is the vorticity magnitude and y^+ is the dimensionless distance to the wall. In the outer region, eddy viscosity is given by $(\nu_t)_{outer} = KC_{cp}F_{wake}F_{Kleb}$, where K and C_{cp} are model constants, $F_{wake} =$ $min(y_{max}F_{max}, C_{wk}y_{max}U_{dif}^2/F_{max})$, and $F_{Kleb} = [1 +$ $5.5(C_{Kleb}y/y_{max})^6]^{-1}$. The F_{max} and y_{max} are determined by the value and corresponding location, respectively, of the maximum of $F=y|\omega||1$ $exp(-y^+/A^+)].$ The quantity U_{dif} is the difference between maximum and minimum velocity magnitudes in the profile and is expressed as $U_{dif} =$ $(U^2+V^2+W^2)_{max^-}^{1/2}(U^2+V^2+W^2)_{min}^{1/2}$. C_{Kleb} and C_{wk} are additional model constants. Numerical values for the model constants are $A^+=26$, $\kappa=0.4$, K=0.0168. $C_{cp}=1.6, C_{wk}=1.0, \text{ and } C_{Kleb}=0.3.$

2.3 A modified Baldwin-Lomax model

Himeno³⁾ proposed a modification of the Baldwin-Lomax model, in order to include anisotropy of turbulence field and effects of pressure gradient in the flow. Details are described in the following.

The non-dimensional Reynolds stress $\overline{u_i u_j}$ equations for unsteady, three-dimensional flow can be written as

$$\frac{\partial \overline{u_i u_j}}{\partial t} + \sum_{l=1}^3 U_l \frac{\partial \overline{u_i u_j}}{\partial x^l}$$
$$= \sum_{l=1}^3 (-\overline{u_j u_l} \frac{\partial U_i}{\partial x^l} - \overline{u_j u_l} \frac{\partial U_i}{\partial x^l})$$

$$-\frac{2}{\operatorname{Rn}}\frac{\overline{\partial u_{i}}}{\partial x^{j}}\frac{\partial u_{j}}{\partial x^{i}} + \overline{p(\frac{\partial u_{i}}{\partial x^{j}} + \frac{\partial u_{j}}{\partial x^{i}})} \\ -\sum_{l=1}^{3}\frac{\partial}{\partial x^{l}}\left\{\overline{u_{i}u_{j}u_{l}} + \delta_{il}\left(\overline{pu_{i}} + \overline{pu_{j}}\right) \\ -\frac{1}{\operatorname{Rn}}\frac{\partial\overline{u_{i}u_{j}}}{\partial x^{l}}\right\}$$
(8)

If we consider only steady flow and balance between convection and generation, equation (8) becomes

$$\sum_{l=1}^{3} U_l \frac{\partial \overline{u_i u_j}}{\partial x^l} = \sum_{l=1}^{3} \left(-\overline{u_i u_l} \frac{\partial U_j}{\partial x^l} - \overline{u_j u_l} \frac{\partial U_i}{\partial x^l} \right)$$
(9)

or

$$\sum_{l=1}^{3} U_{l} \frac{\partial \overline{u_{i} u_{j}}}{\partial x^{l}}$$

$$= -\overline{u_{i} u_{i}} \frac{\partial U_{j}}{\partial x^{i}} - \overline{u_{j} u_{j}} \frac{\partial U_{i}}{\partial x^{j}}$$

$$- \overline{u_{i} u_{j}} (\frac{\partial U_{i}}{\partial x^{i}} + \frac{\partial U_{j}}{\partial x^{j}} + \frac{\partial U_{k}}{\partial x^{k}})$$

$$- \overline{u_{i} u_{k}} \frac{\partial U_{j}}{\partial x^{k}} - \overline{u_{j} u_{k}} \frac{\partial U_{i}}{\partial x^{k}} - \overline{u_{i} u_{j}} \frac{\partial U_{k}}{\partial x^{k}}$$
(10)

where (i, j, k) = (1,2,3) are cyclic order. The third term in the above equation is zero due to the continuity. Assuming the forth through sixth terms to be negligible, and $\overline{u_i u_i} \approx \overline{u_j u_j} \approx \overline{u_k u_k} \approx 2k/3$, where k is turbulent kinetic energy, we get

$$\sum_{l=1}^{3} U_l \frac{\partial \overline{u_i u_j}}{\partial x^l} \propto -\frac{2}{3} k \gamma_{ij} \tag{11}$$

or

$$\overline{u_i u_j} \propto -\frac{2}{3} k \frac{Lt}{Ut} \gamma_{ij} \tag{12}$$

with

$$q_{ij} = \frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i}$$
(13)

where Ut and Lt are characteristic velocity and length scales in the turbulence fields. In similar manner, the expression of $\overline{u_i u_i}$ can be obtained as follows:

$$\sum_{l=1}^{3} U_l \frac{\partial \overline{u_i u_i}}{\partial x^l}$$

-136-

$$= -2\overline{u_i u_i} \frac{\partial U_i}{\partial x_i} - 2\overline{u_j u_j} \frac{\partial U_i}{\partial x_j} - 2\overline{u_i u_k} \frac{\partial U_i}{\partial x_k}$$

$$\approx -\overline{u_i u_j} \gamma_{ij} - \overline{u_i u_k} \gamma_{ik} - \frac{2}{3} k \gamma_{ii}$$
(14)

or

$$\overline{u_i u_i} \propto \frac{Lt}{Ut} \left[-\overline{u_i u_j} \gamma_{ij} - \overline{u_i u_k} \gamma_{ik} - \frac{2}{3} k \gamma_{ii} \right]$$
(15)

and

$$k \propto \frac{Lt}{Ut} \left[-\overline{u_i u_j} \gamma_{ij} - \overline{u_j u_k} \gamma_{jk} - \overline{u_k u_i} \gamma_{ki} \right]$$
(16)

Equations (12), (15) and (16) can be rewritten by using constant c to yield

$$\overline{u_i u_j} = -ck\gamma_{ij} \qquad (i \neq j) \qquad (17)$$

$$\overline{u_i u_i} = -ck\gamma_{ii} + \frac{1}{c\gamma^2} \left[-\overline{u_i u_j} \gamma_{ij} - \overline{u_i u_k} \gamma_{ik} \right]$$
(18)

$$k = \frac{1}{c\gamma^2} \left[-\overline{u_i u_j} \gamma_{ij} - \overline{u_j u_k} \gamma_{jk} - \overline{u_k u_i} \gamma_{ki} \right]$$
(19)

with

$$\gamma = \sqrt{\gamma_{ij}^2 + \gamma_{jk}^2 + \gamma_{ki}^2} \tag{20}$$

In equations (18) and (19), (i, j, k) = (1,2,3) are cyclic order. If we apply and expand the concept of Bradshow for one equation model to three-dimensional flow to get

$$\left|\overline{u_i u_j}\right| = \alpha k, \quad \alpha \approx 0.3 \tag{21}$$

where α is a model constant determined by the experiments, c can be related to α in equation (21) and eddy viscosity ν_t as follows:

$$c = \frac{\alpha}{\gamma}, \quad \nu_t = \frac{\alpha k}{\gamma} = kc$$
 (22)

Finally we have the following algebraic expressions for $\overline{u_i u_j}$, which include anisotropy of turbulence field:

$$-\overline{u_i u_j} = \nu_t \gamma_{ij} - \delta_{ij} k_i$$

$$k_i = k (\gamma_{ij}^2 + \gamma_{ik}^2) / \gamma^2,$$

$$k = \nu_t \gamma / \alpha$$
(23)

where (i, j, k) = (1,2,3) are cyclic order, k is turbulent kinetic energy, which does not appear in the Baldwin-Lomax model expression, and α (=0.4) is a model constant determined by the numerical tests as described in the following section.

On the other hand, Himeno³⁾ also applied the Clauser fitting to eddy viscosity ν_t to account for effects of pressure gradient in the flow, i.e.,

$$\overline{\nu_t} = \nu_t - \frac{\delta^2}{2U_e} \frac{\partial p}{\partial x}$$

$$= K\delta^* U_e - \frac{\delta^2}{2U_e} \frac{\partial p}{\partial x}$$

$$= K\delta^* U_e (1 - \frac{\delta^2}{2K\delta^* U_e^2} \frac{\partial p}{\partial x}) \qquad (24)$$

where K=0.0168 is the Clauser constant, δ and δ^* are boundary layer and displacement thicknesses, and U_e is edge velocity of boundary layer. δ^* and U_e can be replaced by y_{max} and F_{max} in similar manner of Baldwin and Lomax, hence we have the following equation to modify the eddy viscosity based on pressure gradient in streamwise direction s:

$$\overline{\nu_t} = K C_{cp} y_{max} F_{max} \tag{25}$$

$$C_{cp} = C_{cp0} \left\{ 1 - \tanh\left(\beta \frac{y_{max}}{F_{max}^2} \frac{\partial p}{\partial s}\right) \right\}$$
(26)

where $C_{cp0} = 1.6$ and β (=100) are model constants. β is determined by the numerical tests as is done for α . In the present study, $\overline{\nu_t}$ is used instead of ν_t in equation (23) in outer region. This model may be capable for practical use under consideration of its theoretical limitation.

3. Computational method

The unsteady RaNS and continuity equations are solved for the mean-velocity components (U, V, W), pressure p, and eddy viscosity ν_t . The transport equations for $(U, V, W, k, \varepsilon)$ are written in Cartesian coordinates (X, Y, Z) in the physical domain and partially transformed into numerically generated boundary-fitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) . Variables are non-dimensionalized using the freestream velocity U_0 (=1), body length L (=1), and density ρ . The transformed equations are solved using a regular grid, finite-analytic discretization, a PISO-type velocity-pressure coupling algorithm, and the method of lines. Although the present



Fig. 1 Definition sketch of coordinate system solution domain and boundaries.



Fig. 2 Computational grid: (a) body surface; and (b) body crossplane (X=0.8).

solutions are for steady flow, the equations are solved in unsteady form with time serving as a convergence parameter. For more detailed equations, solution procedure and references, see Tahara et al. $^{(4)8)9}$.

Consider a ship fixed in the uniform onset flow $U_0 = (U_0, 0, 0)$ as depicted in Fig. 1. Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward. The solution domain is also shown in the figure. In terms of the notation of Fig. 1, the boundary conditions are described for the original Baldwin-Lomax model approach as follows: on the inlet plane Si, the boundary conditions are provided on the basis of the freestream values, i. e., U=1, V=W=p=0; on the exit plane Se, axial diffusion is negligible, i.e., $\partial^2(U, V, W)/\partial X^2 = \partial p/\partial X = 0$; on the symmetric

plane Sc, the conditions are $\partial(U, W, p)/\partial Y = V = 0$; on waterplane Sw, the conditions are similar to those on Sc, i.e., $\partial(U, V, p)/\partial Z = W = 0$; and on the outer boundary So, the conditions are U=1, W=p=0, and V is determined by the solution to satisfy the continuity equation. On the body surface Sb, the boundary conditions used are $U=V=W=\partial p/\partial n=0$ (where n is normal to the body).

In the present study, the computation was first performed with the Baldwin-Lomax model, and took over with the two-layer $k - \varepsilon$ and modified Baldwin-Lomax models from the midship section (X=0.5) using the Baldwin-Lomax model results as an initial guess. The boundary conditions for the two-layer $k - \varepsilon$ model approach are similar to those of the Baldwin-Lomax model approaches, but conditions for k and ε are included as follows: on the inlet plane Si (midship section), the boundary conditions for U, V, W, and p are provided from the Baldwin-Lomax results, and conditions for k and ϵ are $\partial(k,\epsilon)/\partial X=0$; on the exit plane Se, axial diffusion is negligible for turbulent quantities, i.e., $\partial^2(k,\varepsilon)/\partial X^2=0$; on the symmetric plane Sc, the conditions are $\partial(k,\varepsilon)/\partial Y=0$; on waterplane Sw, the conditions are similar to those on Sc, i.e., $\partial(k,\varepsilon)/\partial Z=0$; and on the outer boundary So, the conditions are $\partial(k,\varepsilon)/\partial r=0$ (where r is the radial direction as shown in Fig. 1). On the body surface Sb. the boundary conditions used are k=0 and the condition for ε is algebraically given. For the modified Baldwin-Lomax model approach, the boundary conditions are similar to those of the Baldwin-Lomax model approach; however the inlet conditions for the velocity and pressure fields are same as those of the two-layer $k - \varepsilon$ model approach.

In this study, the initial conditions are taken from the freestream values, i. e., U=1, V=W=p=0; and very small values are specified for k and ε . It is assumed that the flow is already turbulent upstream of the bow.

4. Results and discussion

As already indicated, the computational results are presented for the SR196a tanker form for Rn=4,000,000. The turbulence models employed in the computations are the original Baldwin-Lomax model⁹⁾, the two-layer k- ε model used by Tahara⁴⁾, and the modified Baldwin-Lomax model proposed by Himeno³⁾ (referred to as B-L model, two-layer model, and modified B-L model, respectively).

The computational grids were obtained using the elliptic-algebraic method of Tahara $^{(4)9)}$. The inlet, exit and outer boundaries are located at X=(-0.4, 2.0) and r=1, respectively. The first grid point off the body surface is located in the viscous sublayer, i. e., $y^+ < 0.2$, where $y^+=RnU_\tau y=Rn\sqrt{\tau_w/\rho U_0^2}y$ is the dimensionless distance, and τ_w is the wall-shear stress. Figs. 2-a and -b show the computational grid on the body surface and crossplane at X=0.8, respectively. For two-layer k- ε model approach, the matching boundary between the near-wall and outer layers is located in 50 $< y^+ < 100$, where the location was initially guessed, then updated several times until the requirement was satisfied.

The numbers of grids used are 85x30x20 (=51,000) in the axial, radial, and circumferential directions, respectively. Note that nearly half of the original grids were used in the two-layer and modified B-L model approaches, since actual computational domain is smaller as the above mentioned. The supercomputer (SX-3R) time and central memory were about 0.5 hours and 30 MB for the B-L model approach, and 0.25 hours and 15MB for the two-layer and modified B-L model approaches. The convergence criterion was that the change in solution be less than about 0.05% for all variables, which was satisfied within 500 global iterations for all turbulence models.

For the SR196a tanker form, detailed measurements on the mean-velocity and Reynolds-stress fields were made in Osaka University wind tunnel, and currently data for stern region are available for comparison (Suzuki et al. ³⁾, average Rn=4,000,000; referred to as OSAKA). Although the present computations cover entire ship hull, the discussions to follow are focused on the stern region near the propeller disk, which is of great interest in engineering purpose.

Figs. 3 through 8 show the crossplane vectors (V-W), the axial-velocity (U) contours, and Reynolds stress contours for X=(0.95, 0.988). Figs. 9 and 10 provide detailed comparison of the velocity and Reynolds stress profiles between the computations and the experiments for X=0.988. The computed surface pressure (Cp=p/.5) contours, the normalized shear stress vectors, and the pressure (Cp) and the axial shear stress $(C_{\tau x}=\tau_{wx}/.5\rho U_0^2)$ profiles are compared in Figs. 11 and 12. Lastly, Fig. 13 provides comparison of resistance coefficients. In the presentation of the results to follow, variables are non-dimensionalized using the ship length L, the free-stream speed U_0 and fluid density ρ .





4.1 X=0.95

At this station, the experimental data indicate detailed characteristics of the thick boundary layer near the stern. The U contours of the measurements (Fig. 3-a) clearly show a pronounced bulge in the boundary layer near the region of maximum hull concavity and reduction in the boundary layer near the centerplane. The U contours also display complex aspects in the inner region near the bilge, where the contours already



Fig. 4 Comparison of Reynolds stress contours at X=0.950: (a) measurements; (b) two-layer k- ε model; and (c) modified Baldwin-Lomax model; columns, $\overline{uu}, \overline{vv}$, and \overline{ww} contours, respectively.



Fig. 5 Comparison of Reynolds stress contours at X=0.950: (a) measurements; (b) two-layer $k \cdot \varepsilon$ model; (c) modified Baldwin-Lomax model; and (d) Baldwin-Lomax model; columns, $\overline{uv}, \overline{uw}$, and \overline{vw} contours, respectively.

start to form the "hook shape." The V-W vectors of the measurements are directed upward and towards the centerplane, and finally downward near the hull, where the stern-bilge vortex is clearly observed. The measured $\overline{uu}, \overline{vv}$, and \overline{ww} contours (Fig. 4-a) show similar extents as those of the U contours, but with reverse trend in magnitude. The measured $\overline{uv}, \overline{uw}$, and \overline{vw} contours (Fig. 5-a) also show many complexities near the "hook shape." Those aspects of the Reynolds stresses can be related to the shear flow structure affected by the stern bilge vortex. In order to capture the origin of the stern-bilge vortex, which is shown in the experimental data, the computational approaches must accurately reproduce the Reynolds stress fields as well as the mean velocity fields.

The above-mentioned are generally reproduced in the present computations with the two-layer and modified B-L models, regarding the U contours and V-Wvectors (Figs. 3-b and -c). However, the B-L model results (Fig. 3-d) completely lack details, in which the "hook shape" in the measured U contours are obviously missing, and the magnitudes of V-W vectors are smaller than those of the measurements. The differences of the U contours between the two-layer and modified B-L model results are also observed, i.e., the values of the two-layer results are slightly smaller in the region very close to the hull, especially near the bilge. The "hook shape" U contours of the two-layer results have somewhat larger extents than those of the measurements, which may be related to the underprediction of the U magnitude in the region. The modified B-L model did not yield clear "hook shape" U contours, but predicts closer values to those of the measurements than the two-layer model. The Reynolds stress distributions $\overline{uu}, \overline{vv}$, and \overline{ww} (Figs. 4-b and c) are generally well reproduced by the two-layer and modified B-L model approaches, i.e., the location and magnitude of the high value regions are similar to those of measurements; however, the details in the measurements are absent in the computations, e.g., the extent of relatively high values near the waterline are not completely reproduced.

In contrast, the present turbulence models show some difficulties in predicting the $\overline{uv}, \overline{uw}$, and \overline{vw} distributions, i.e., complexities displayed in the measurements are completely missing, although predicted values near the bilge is similar to those of the measurements. These difficulties are likely due to the anisotropy of turbulence fields, i.e., more advanced turbulence model may be able to yield better resolution. As clearly shown in the figure (Fig. 5-d), the B-L model considerably overpredicts the magnitude and extent of the Reynolds stress distributions.



Fig. 6 Comparison of solutions at X=0.988: (a) measurements; (b) two-layer k-ε model;
(c) modified Baldwin-Lomax model; and
(d) Baldwin-Lomax model; columns, crossplane-velocity vectors; and axial-velocity contours, respectively.

4.2 X=0.988

At this station, the U contours of the measurements (Fig. 6-a) display merging of the boundary layer into the wake and its initial evolution. The measured V-Wvectors (Fig. 6-a) show the flow is upward and towards the centerplane, and finally directed downward at the centerplane, where the stern-bilge vortex evolves into



Fig. 7 Comparison of Reynolds stress contours at X=0.988: (a) measurements; (b) two-layer k- ε model; and (c) modified Baldwin-Lomax model; columns, $\overline{uu}, \overline{vv}$, and \overline{ww} contours, respectively.



Fig. 8 Comparison of Reynolds stress contours at X=0.988: (a) measurements; (b) two-layer k- ε model; (c) modified Baldwin-Lomax model; and (d) Baldwin-Lomax model; columns, $\overline{uv}, \overline{uw}$, and \overline{vw} contours, respectively.

a longitudinal vortex. The measured U contours display a pronounced bulge in the boundary layer in outer region, and complexities including the "hook shape" near the centerplane at large depth. These flow features are clearly observed in the velocity profiles (Fig. 9). At middle depths, the measured U profiles indicate the smallest values in the region slightly away from the centerplane, which is related to the abovementioned "hook shape." The V profiles of the measurements are mostly negative at smaller depths, and positive near the centerplane at larger depths. The measured W profiles are mostly positive, and negative with increased magnitude near the centerplane at middle depths. Such complex flow aspects are due to the stern-bilge vortex.

As is the case for the previous station, the measured $\overline{uu}, \overline{vv}$, and \overline{ww} contours (Fig. 7-a) show similar extents as those of the U contours, but with reverse trend in magnitude. The measured $\overline{uv}, \overline{uw}$, and \overline{vw} contours (Fig. 8-a) also display similar extents as those of the U contours, but the distribution of positive and negative values are rather complicated, which is correlated with the structure of shear flow. Suzuki et al.³⁾ pointed out the anisotropy in the Reynolds stress fields in this station, i.e., assumption of isotropic eddy viscosity may no longer be reliable. The challenge for the numerical approaches is to resolve the growth of the stern-bilge vortex and the mean streamwise velocity field as well as their interaction with the Reynolds stress fields.

The U contours of the B-L model results (Fig. 6-d) do not display the "hook shape" and completely lack details, which is also clearly observed in the U profiles (Fig. 9). The V-W vectors of the B-L model results (Fig. 6-d) have smaller magnitudes than those of the measurements, in which the V and W profiles (Fig. 9) display clear discrepancies between the results and the measurements. Significant improvements are observed in the two-layer and modified B-L model results (Figs. 6-b and -c) regarding the U contours and the V-W vectors. The U contours of the two-layer results show many similarities to those of the measurements; however, the extent of the "hook shape" is slightly shifted towards the centerplane, and the values near the center of the "hook shape" are somewhat underpredicted. The U contours of the modified B-L model results show similar aspects of the two-layer model results, but the values near the "hook shape" are closer to those of the measurements. The V-W vectors of the two-layer and modified B-L model results display larger magnitudes than those of the B-L model results, which are more similar to those of the measurements.

The above-mentioned is consistent with the velocity profiles (Fig. 9), i.e., the two-layer and modified B-L model results show quite a few improvements over the B-L model results. However, the two-layer model slightly underpredicts and overpredicts the values of the U profiles near the centerplane at most depths, and at the largest depth, respectively. The U profiles of the modified B-L model results show closer agreements with the measurements, but details displayed in the measurements are still absent. The Vprofiles of the two-layer and the modified B-L model results show relatively good agreement with the measurements, where the two-layer model results are in better agreement with the measurements. The W profiles of the two-layer and the modified B-L model results display many similarities to those of the measurements; however, the agreement of the magnitudes is not satisfactory at middle depths in the region slightly away from the centerplane. This may be due to the insufficient grid resolution and accuracy of turbulence models, or possibly due to the wall effects in the wind tunnel experiments, which are not considered in the present computations.

As compared to the resolution of mean-velocity fields, those of Reynolds stress fields are not satisfactory in the present two-layer results. The two-layer approach mostly underpredicts Reynolds stresses (Figs. 7-b and 8-b), which may be due to the inadequate ε boundary condition in the wake region, which is one of the well-known issues of the k- ε model. In the present computation, a simple symmetric condition is imposed for ε at the wake centerplane, in which near wall effects might be important in the region very close to the stern. This issue will be considered in more detail in the future work, including the modification of computational grid and its resolution. On the other hand, the modified B-L model results indicate similar Reynolds stress magnitudes as those of the measurements (Figs. 7-c and 8-c), especially for $\overline{uu}, \overline{vv}$, and \overline{ww} , which appear to be better agreement than those of the two-layer results. As noted by others¹⁾²⁾³⁾, clear overpredictions of the magnitude are observed in the B-L model results.



Fig. 9 Comparison of velocity profiles at X=0.988.



Fig. 10 Comparison of Reynolds stress profiles at X=0.988.



Fig. 11 Comparison of solutions on hull surface: (a) Baldwin-Lomax model; (b) two-layer k- ε model; and (c) modified Baldwin-Lomax model; columns, Cp contours; and normalized shear stress vectors, respectively.



Fig. 12 Comparison of surface pressure and shear stress profiles: (a) Cp profiles; and (b) $C_{\tau x}$ profiles; columns, keel; midgirth; and waterline, respectively.



Fig. 13 Comparison of resistance coefficients.

4.3 Influences of α and β on the Reynolds stresses

As mentioned before, the model constants α and β in the present modified B-L model were determined to reproduce the mean-velocity and Reynolds stress fields of the measurements at X=0.988, i.e., the location of the propeller disk. Fig. 10 shows influences of α and β on the Reynolds stress solutions. As seen in the figure, α mainly influences the magnitude of the $\overline{uu}, \overline{vv}$, and \overline{ww} distributions, and β global profiles of the all Reynolds stresses. As α decreases, magnitudes of the $\overline{uu}, \overline{vv}$, and \overline{ww} increase. As β decreases, the distributions of Reynolds stresses generally tend to be flat. Note that $\beta < 10$ yields quite plane distribution of the U profiles, where the "hook shape" U contours totally disappear. It must also be noted that smaller α less than 0.2 breakdown the computation due to considerably steep distribution of the resultant $\overline{uu}, \overline{vv}$, and \overline{ww} . In the present study, finally $\alpha = 0.4$ and $\beta = 100$ were selected under consideration of global agreement between the solutions and experimental data.

4.4 Surface pressure and shear stresses

The computed surface pressure and normalized shear stress vectors are compared in Fig. 11. Differences of the Cp contours are highlighted near the stern, which correlate the clear differences of the normalized shear stress vectors. Those differences are obvious in the region near the stern and bilge, where the vectors of the two-layer and modified B-L model results are directed upward and downward in more complicated manner than those of the B-L model results. The above mentioned are consistent with the Cp and $C_{\tau x}$ (X component of the shear stress vector) profiles shown in Fig. 12. The differences of the Cpprofiles among three results are relatively small along keel and waterline, but clear along midgirth near the stern. On the other hand, significant differences are observed for $C_{\tau x}$ profiles, i.e., the two-layer and modified B-L model results generally indicate smaller values than the B-L model results. The girthwise shear stress of the modified B-L model results sometimes jumps, which may be caused by a mismatching with the B-L model at the midship section.

Integration of the pressure and shear stress distributions over the hull surface yields the pressure and frictional resistance, respectively, whose coefficients are shown in Fig. 13. Note that the pressure and frictional parts of the resistance are denoted as CPand CF, respectively. In the figure, the modified B-L model results of $(\alpha, \beta) = (0.3, 100; \text{ Model B})$ and $(\alpha, \beta) = (0.3, 500; \text{ Model C})$ are included in addition to results of $(\alpha, \beta) = (0.4, 100; \text{ Model A})$, i.e., the present model. It appears that the CF of the computations are all smaller than Schoenherr's, while the results of the modified B-L model $(\alpha, \beta) = (0.4, 100; \text{ Model})$ A) is the closest. The differences of CF can be related to the $C_{\tau x}$ distribution discussed above, more specifically, direction of the shear-stress vectors. Further study is needed to validate the results including more detailed comparison with available experimental data or application of the present turbulence models to other tanker forms.

5. Concluding remarks

This work presents applications of isotropic and anisotropic turbulence models in computation of ship boundary-layer and wake flows. The fully-elliptic RaNS and continuity equations are solved with several kinds of turbulence models, i.e., the original and modified Baldwin-Lomax models, and the two-layer k- ε model. The numerical results are obtained, using a regular grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. The results are presented and discussed for the SR196a tanker form for zero Froude number, including detailed comparisons with the measurements in order to validate the present computational approach and investigate the relative performance of the turbulence models.

In this paper, comparisons with the measurements are presented only for the stern and near-wake region, which is due to the limited experimental data. The Baldwin-Lomax model indicates many difficulties in reproducing the complex flow features in the region,

which may be due to the limitation of its underlying assumptions. The two-layer $k - \varepsilon$ and the modified Baldwin-Lomax model results show clear improvements of the solutions regarding the mean-velocity fields. Although the present two-layer $k - \varepsilon$ model predicts general feature of Reynolds stress fields near the hull, some difficulties are observed in the near wake region. The modified Baldwin-Lomax model yields better Reynolds stress fields, regarding the magnitude and location of high values; however, details displayed in the measurements are not correctly reproduced. More advanced turbulence models may be able to yield better resolution. The frictional resistance coefficients of the three turbulence models are all smaller than the value of Schoenherr, where further study is needed to validate the results including more detailed comparison with available experimental data.

Finally, the issues that need to be addressed while further developing and validating the present approaches are summarized as follows: further validation of the present methods through more detailed comparisons with available experimental data and application to other tanker forms; further evaluation of the present turbulence models in conjunction with the extension to the higher-order models; and modification of the numerical method regarding the computational grid and the solution algorithm for higher accuracy and efficiency. Also it is of interest to include free-surface effects through the extension demonstrated by Tahara and Stern⁹.

References

- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.
- Proc. Workshop on Wave Resistance and Viscous Flow, Tokyo, July 1994.
- SR222 2nd Report, March 1995 (unpublished) [Japanese].
- Tahara, Y., "An Application of Two-Layer k-ε Model to Ship Flow Computation," to appear J. Society of Naval Architects of Japan, 1995.
- 5) Iacovides, H. and Launder, B.E., "The Numerical Simulation of Flow and Heat Transfer in Tubes in Orthogonal-Mode Rotation," Proc. 6th symp. Turbulent Shear Flows, Toulouse, France, 1987.
- 6) Chen, H.C. and Patel, V.C., "The Flow Around Wing-Body Junctions," Proc. 4th Symp. Num.



Fig. 14 Surface Cp contours and normalized shear stress vectors for modified Baldwin-Lomax model $(\alpha, \beta) = (\infty, 100)$.

Phys. Aspects Aerodyn. Flows, Cebeci ed., Long Beach, CA, 1989.

- Rodi, W. and Scheuerer, G., "Scrutinizing the k-ε Turbulence Model under Adverse Pressure Gradient Conditions," ASME J. Fluids Eng., Vol. 108, 1986, pp. 174-179.
- Tahara, Y., "Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments," J. Kansai Society of Naval Architects, Japan, No. 220, September 1993, pp. 29-47.
- 9) Tahara Y. and Stern, F., "A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.

Discussion

[Discussion] (大阪大学) 鈴木 敏夫

簡便な乱流モデルによる計算結果の比較をお示しい ただき有難うございます.Fig.11 について質問させて いただきます.船体中央部近傍ではガース方向の圧力 変化は小さいと思われます。その立場で(C)図を見る と、Midship以降急激に壁面流線がビルジ部に寄って きていますが、その原因についてお教えください.な んとなくデータの配列まちがいのようにも思えますの でチェックの方もよろしくお願いします.

[Author's Reply]

ご指摘有難うございます.確かに Midship において Mod.B-L Model の壁面摩擦応力ベクトルには不連続 性があり、またそれ以降もビルジ部へ偏る傾向を見せ ています.その理由は Midship における乱流モデル の不連続性にあると思われます.今回ご紹介した研 究では Midship で計算空間を分離し、その前半部を B-L Model で、後半部を他の乱流モデルで計算してい ます.今回提案する Mod.B-L Model には (23) 式に示 される乱動エネルギー (k)の項があり、これは計算上 (2) 式の圧力項に加えられる形になります. ビルジ部 Midship 近傍の表面圧力分布に関しては B-L Model お よび Mod.B-L Model の解に差が小さいことから、(23) 式の k項について Midship における不連続が生じたも のと思われます. Fig.14 に示された結果は (23) 式から k項を除いた形、すなわち $\alpha \rightarrow \infty$ とした場合のもので す. Midship およびそれ以降の壁面摩擦応力ベクトル 場の急変も見られなくなります. 従って、単一計算空 間を用い、船体前半部から Mod.B-L Model を用いれ ば不連続性に関する問題はなくなると思われます。

A Computational Study of Three-Dimensional Laminar Separation on a Prolate Spheroid at Incidence *1

Yusuke TAHARA (Member) *2 , Satoshi MITARAI (Student member) *2

 and

Yoji HIMENO (Member)*2

This paper presents a computational study of three-dimensional laminar separation on a prolate spheroid at incidence. The fully-elliptic Navier-Stokes and continuity equations are solved using the regular grid, finite-analytic discretization, a PISO-type velocity-pressure coupling algorithm, and method of lines. An O-O-type body-conforming grid, which is suitable for the detailed study of the present problem, is used in the computation. An overview is given of the present approach, and numerical results are presented and discussed for a 6:1 prolate spheroid for several angles of attack. The latter includes comparisons with available experimental data. The present computational results display detailed flow structures in the post-separation region and near wake, such as the generation and development of the girthwise and longitudinal vortices or the separation patterns which can not be classified by the conventional open- and closed-type separation models.

Keywords : Laminar Separation, Navier-Stokes Equation, Boundary Layers and Wakes, Spheroid, Finite-Analytic Scheme

1. Introduction

An important factor in the design of ships, airplanes, automobiles, etc., is accurate prediction and control of flow separation. It is a classical and important problem in fluid-dynamics or aerodynamics related engineering. The flow separation generally results in an increase in drag, a loss of lift, and often in self-induced unsteadiness, therefore the designers are usually concerned with avoiding or controlling it. Initially, the interest was primarily with experimental study based on visualization of the limiting or frictional streamlines on the body surface, and computational study based on the threedimensional integral and differential boundary-layer equation methods. Nowadays, the emphasis is placed on more detailed investigation of the flows by using the fully-elliptic Navier-Stokes (NS) or Reynolds-averaged

Navier-Stokes (RaNS) equation methods. Recent developments of the NS and RaNS equation methods encourage further detailed investigation of flows in the post-separation region, on which relatively few studies have been done. The present paper is central to the problems, i.e., it concerns a computational study of the three-dimensional laminar boundary layers and wakes involving separation around a prolate spheroid at incidence with main emphasis on the flow structure in the post-separation region and near wake.

Maskell¹⁾ classified the three-dimensional separation into two types, i.e., the bubble and free-vortex layer types. As is well known, the bubble formation requires the existence of a saddle point, while the free-vortex-layer separation involves only ordinary points. Therefore these are often called the singular and ordinary separations, respectively. Legendre²⁾ and Lighthill³⁾ proposed a more mathematical and rigorous definition of the three-dimensional flow separation by postulating that the frictional streamlines constitute a continuous vector field. Wang⁴⁾ took over the Maskell's idea, but introduced the concept of open

^{*1} Read at the Joint Autumn Meeting of Three Societies of Naval Architects in Japan, November 16, 1995, Received December 1, 1995

^{*2} Osaka Prefecture University

and closed separations based on the accessibility or inaccessibility of the separation region from upstream, which is pointed out by Legendre or Lighthill. Tobak and Peake⁵⁾ suggested another way to tell the two separation types, i.e., the local and global separations, based on the topological structure and structural stability given by the theory of ordinary differential equations. Most of recent studies have applied the concept of open and closed separations, because of its simplicity and adequacy to describe various flow patterns.

Historically, a prolate spheroid has been often used to study three-dimensional separation. Even though the geometry is relatively simple, a variety of separation patterns can be observed by changing the angle of attack. Most of the experimental studies are based on flow visualizations. Han and Patel⁶⁾ first conducted systematic experiments, where the observed separation patterns were consistent with the open and closed separation models proposed by Maskell and Wang. Werle⁷) made a series of visualizations to study separation patterns on a family of ellipsoids of different axes ratios and at different Reynolds number (Re). Costisetal.⁸⁾ concluded that an open-type separation prevails from moderate to high incidence. Nishio⁹⁾¹⁰⁾ observed several separation patterns on a 6:1 prolate spheroid, and made detailed measurements on the mean velocity and pressure fields as well as the surface pressure distributions. The most recent experiments done by Wang et al.¹¹⁾ claimed that flow separation changes from the closed type to the open type and back to the closed type again as the incidence increases.

The computational studies were initially based on the solution of integral and differential boundary-layer equations, and more recently the NS and RaNS or some reduced forms of those equations. $Wang^{(2)-14)}$ first presented computational study on a 4:1 prolate spheroid at incidence, using a boundary-layer equation method. Hayashita and Himeno¹⁵⁾, Patel and Choi¹⁶⁾, Cebeci et al.¹⁷⁾¹⁸⁾ and Patel and Baek¹⁹⁾ took over the study by using the boundary-layer equation methods, where very detailed investigations had been completed in the pre-separation region. Recent studies based on NS equation methods $^{20)-27)}$ have overcome the limitation of boundary-layer equation methods, i.e., the computation can not be continued in the postseparation region. However, most of the studies have emphasized the numerical aspects of the calculations

rather than the physics of separation. As an exception, Kim and $Patel^{26}-27$ systematically investigated the global separation patterns for various incidence angles; however, detailed investigation of the flow structure in the post-separation region and near wake had not been completed. Hence the present study was motivated with the modification and extension of the recently developed RaNS equation methods²⁸⁾⁻³¹⁾ for the ship flow to the present problem.

This paper presents a computational study of threedimensional laminar separation on a prolate spheroid at incidence with main emphasis on the flow structure in the post-separation region and near wake. A numerical method is developed for the present study, based on extensions and modifications of the method of Tahara²⁸⁾⁻³¹⁾. The fully-elliptic Navier-Stokes and continuity equations are solved using the regular grid, finite-analytic discretization, a PISO-type velocitypressure coupling algorithm, and method of lines. An O-O-type body-conforming grid, which is suitable for the detailed study of the present problem, is used in the computation. An overview is given of the present approach, and numerical results are presented and discussed for a 6:1 prolate spheroid for several angles of attack. The latter includes comparisons with available experimental data. The present computational results display detailed flow structures in the post-separation region and near wake, such as the generation and development of the girthwise and longitudinal vortices or the separation patterns which can not be classified by the conventional open- and closed-type separation models. Lastly, some concluding remarks are made regarding the requirements and prognosis for further study, which includes turbulent transition and separation problem.

2. Computational method

The non-dimensional NS equations for unsteady, three-dimensional incompressible flow can be written in Cartesian tensor notation as

$$\sum_{i=1}^{3} \frac{\partial U_i}{\partial x^i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \sum_{j=1}^3 \left(U_j \frac{\partial U_i}{\partial x^j} \right) = \frac{\partial p}{\partial x^i} - \frac{1}{\operatorname{Re}} \nabla^2 U_i = 0$$

with
$$\nabla^2 = \sum_{j=1}^3 \frac{\partial^2}{\partial x^j \partial x^j}$$
 (2)

where $U_i = (U, V, W)$ are the Cartesian components of mean velocities normalized by the reference velocity $U_0, x_i = (X, Y, Z)$ are the dimensionless coordinates normalized by a characteristic length L, p is the pressure normalized by ρU_0^2 , $Re=U_0L/\nu$ is the Reynolds number, ρ is the fluid density, and ν is the kinematic viscosity.

The transport equations for (U, V, W) are partially transformed into numerically generated boundaryfitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) , then solved with continuity equation using a regular grid, finite-analytic discretization, a PISO-type velocity-pressure coupling algorithm, and the method of lines. Although the present solutions are for steady flow, the equations are solved in unsteady form with time serving as a convergence parameter. For more detailed equations, solution procedure and references, see Tahara et al.²⁸⁾⁻³¹.



Fig. 1 Definition sketch of coordinate system solution domain and boundaries.

Consider a spheroid fixed in the uniform onset flow $U_0 = (U_0 \cos \alpha, 0, U_0 \sin \alpha)$ as depicted in Fig. 1 (where α is the angle of attack, and taken to be positive for the onset flow with positive component in the Z direction). Take the Cartesian coordinate system with the origin at the center of spheroid, X and Y axes on the horizontal plane, and Z axis directed vertically upward. The vertical plane Y=0 is a symmetry plane and contains the windward and leeward centerlines of the body. The solution domain is also shown in the figure. In terms of the notation of Fig. 1, the boundary conditions on each of the boundaries are described as follows: on outer boundary So, the boundary conditions are provided on the

basis of the freestream values, i. e., $U = U_0 \cos \alpha$, V = 0, $W = U_0 \sin \alpha$ and p is linearly extrapolated from the interior nodes; on exit plane Se, axial diffusion is negligible so that the exit conditions used are $\partial^2(U, V, W)/\partial^2 X = \partial p/\partial X = 0$; on centerplane Sc, the conditions imposed are $\partial(U, W, p)/\partial Y = V = 0$; and on body surface Sb, boundary conditions used are $U = V = W = \partial p/\partial n = 0$ (where n is normal to the body). In this study, the initial conditions are taken from the freestream values.



Fig. 2 Computational grid.

3. Results and discussion

As already indicated, the computational results are presented for a 6:1 prolate spheroid. Nishio et al.¹⁰⁾ observed a variety of separation patterns in the circulation water tank (Re=113,000), which are available for comparison with the present computational results. Nishio et al.¹⁰⁾ also made detailed measurements on velocity and pressure fields as well as surface pressure distribution in Osaka University wind tunnel (Re=2,700,000). Note that the present computations are for laminar flow, although the experimental data for Re=113,000 might be influenced by the turbulent transition and separation. It must also be noted that the present numerical approach is not based on the time accurate scheme, therefore the converged solution may not reproduce all flow features in the separation region, where the natural unsteadiness usually exists due to an oscillation of the separation bubble and possibly vortex shedding at the tail. However, this issue related the unsteadiness will not be further pursued in the present study, but left for the

future work. In the following, computational results are presented and discussed, in which variables are non-dimensionalized using the longitudinal spheroid length L, the free-stream speed U_0 and fluid density ρ .

The computational grids were obtained using the elliptic-algebraic method of Tahara³⁾. The outer boundary is located on a spheroid surface, which is given by $(X/3)^2 + (Y/2)^2 + (Z/2)^2 = 1$. The first grid point off the body surface is located at $y\approx 0.000005$, where y is the distance from the wall.

The numbers of grids used are $180 \times 30 \times 25$ (=135,000) in the axial, radial, and circumferential directions, respectively. Fig. 2 shows the present computational grids. The conditions for the computation were as follows: L=1; $U_0=1$; Re=100,000; and $\alpha =0^{\circ}$, 2° , 4° , 6° , 8° , 10° , and 12° . The values of the time increment $\Delta \tau$ and underrelaxation factors for velocity ($\alpha_U, \alpha_V, \alpha_W$) and pressure (α_p) were as follows: $\Delta \tau=0.01$; $\alpha_U = \alpha_V = \alpha_W = 1$; and $\alpha_p=0.1$. The workstation (HP-9000 712/80) time and central memory were about 111 hours and 64 MB for each incidence angle. The convergence criterion was that the change in solution be less than about 0.05% for all variables, which was satisfied within 1000 global iterations.

Figs. 3-a through -d show the frictional (or limiting) streamlines, the surface pressure $(C_p = p/.5)$ contours, the shear stress $(C_{\tau} = \tau_w/.5\rho U_0^2)$ contours, and the streamwise pressure gradient ($C_{p\bar{\tau}}$ = $\nabla C_p \cdot \bar{\tau}$), where $\bar{\tau}$ is the normalized shear stress vector) contours, respectively, for $\alpha = 0^{\circ}$, 4° , 8° , and 12°. Fig. 4 shows the iso-value surfaces of the magnitude of modified Lam vector $|\mathbf{L}\mathbf{v}^*| = |\mathbf{L}\mathbf{v}|/|\mathbf{U}|$ (where $L_V = \omega \times U, U = (U, V, W)$, and ω is the vorticity vector) for the same α . Fig. 5 compares global separation patterns between observation by Nishio et al.¹⁰⁾ for $\alpha = 11^{\circ}$ and the present solution for $\alpha = 12^{\circ}$. Figs. 6 and 7 provide local views of the frictional streamlines, the normalized shear stress vectors, and zero shear stress contours. Figs. 8-a and -b show local separation patterns observed by Nishio et al.¹⁰) for $\alpha = 4^{\circ}$ and 11°, respectively, and the present results for similar incidence angles are shown in Figs. 9-a and -b. Figs. 10 through 12 provide streamlines on the crossflow planes, and Fig. 13 shows the iso-value surfaces of $\nabla^2 p$ for $\alpha = 4^\circ$. Lastly, Fig. 14 compares the lift and drag coefficients for all α considered in the present computation. Note that X=-0.5 and 0.5 correspond to the nose and tail of the spheroid, respectively; and $\theta = -90^{\circ}$ and 90° the windward and leeward centerlines of the body, respectively.

3.1 Global observation

For $\alpha = 0^{\circ}$, the present solutions appear to be axisymmetric as shown in Fig. 3. Kim²⁷⁾ mentioned about the three dimensionality of the flows near the tail, which is likely due to the numerical errors in the computation; however it is not obvious in the present results. The frictional streamlines (Fig. 3-a, $\alpha = 0^{\circ}$) indicate the closed-type separation near the tail. The C_p contours (Fig. 3-b, $\alpha = 0^\circ$) mostly show negative values except near the nose and tail, where considerably high values are indicated very close to the nose. The C_{τ} contours (Fig. 3-c, $\alpha = 0^{\circ}$) display high values near the nose and the values decrease as X increases in the pre-separation region, which is related to the development of the boundary layer. The streamwise pressure gradient $C_{p\bar{\tau}}$ correlates the acceleration and deceleration of the flow, and developments of the boundary layer. The $C_{p\bar{\tau}}$ contours (Fig. 3-d, $\alpha = 0^{\circ}$) mostly show favorable and adverse pressure gradients in the forward and rear halves of the body, respectively.

For $\alpha = 4^{\circ}$, The results display completely different aspects as compared to those for $\alpha = 0^{\circ}$. A pocket of low pressure region is observed on the forward half of the body in leeward side (Fig. 3-b, $\alpha = 4^{\circ}$), which causes the flow to converge from the windward towards leeward sides. The frictional streamlines (Fig. 3-a, $\alpha = 4^{\circ}$) on the leeward side first converge towards the leeward centerline over the forward half of the body, then diverge over the rear half. The frictional streamlines on the windward side diverge from the windward centerline over most part of the body, except in the rear half of the body near the windward centerline. Finally the frictional streamlines form the characterized separation pattern for this incidence angle, which is found to be the closed-type separation. The C_{τ} contours (Fig. 3-c, $\alpha = 4^{\circ}$) also indicate many differences from those for $\alpha = 0^{\circ}$. The values of C_{τ} are generally higher in the windward side than the leeward side in the pre-separation region, which is related to thicker boundary layer in the leeward side of the body. The streamwise flow is accelerated in the region of negative $C_{p\bar{\tau}}$ contours (Fig. 3-d, $\alpha = 4^{\circ}$), which mostly show favorable and adverse pressure gradients



Fig. 3 Global comparison of solutions: (a) frictional streamlines; (b) C_p contours; (c) C_{τ} contours; and (d) $C_{p\bar{\tau}}$ contours; rows, $\alpha = 0^{\circ}$, 4°, 8°, and 12°, respectively.



Fig. 4 Iso- $|\mathbb{L}_{\mathbf{V}}^*|$ surfaces: (a) $\alpha = 0^\circ$; (b) $\alpha = 4^\circ$; (c) $\alpha = 8^\circ$; and (d) $\alpha = 12^\circ$.

in the forward and rear halves of the body in the preseparation region, respectively. The iso- $|\mathbf{L}_{\mathbf{V}}^*|$ surfaces (Fig. 4) perceptively display the shear flow structure related to the development and separation of boundary layer. The iso- $|\mathbf{L}_{\mathbf{V}}^*|$ surfaces for $\alpha = 4^\circ$ are completely different from those for $\alpha = 0^\circ$, which indicates that the shear flow structure is strongly influenced by changes in the incidence angle.

For $\alpha = 8^{\circ}$ and 12°, the pockets of low pressure region are clearly observed on the forward half of the body in leeward side (Fig. 3-b, $\alpha = 8^{\circ}$ and 12°), and those pockets are narrower and longer than that of $\alpha = 4^{\circ}$ in the Z and X directions, respectively. The value in the pocket decreases as the incidence angle increases from $\alpha = 4^{\circ}$ to 12°. Those configurations of the low pressure pockets cause the flow to converge from the windward towards leeward sides earlier than the case for $\alpha = 4^{\circ}$. Because of the elongate low pressure pocket on the flank of the body, the divergence of the frictional streamlines after the initial convergence on the leeward side starts earlier than that for $\alpha = 4^{\circ}$. The earlier divergence of the frictional streamlines from the leeward centerline results in earlier convergence of the streamlines with which come from the windward side. The merging of the frictional streamlines is the necessary condition for an open separation. The frictional streamlines for $\alpha = 12^{\circ}$ show convergence of the flow in larger extent than that for $\alpha = 8^{\circ}$, i.e., the flows from the windward and leeward sides of the body meet in the leeward and forward side of the body, and the converged streamlines penetrate towards the separation region in the rear part of the body. Along the open separation line, the boundary layer is rolled up into a longitudinal vortices.

The above arguments for $\alpha = 8^{\circ}$ and 12° are supported by the iso- $|\mathbf{L}_{\mathbf{V}}^*|$ surfaces shown in Fig. 4, where a crack of iso-value surface is observed near the convergence of the frictional streamlines. The extents of the C_{τ} contours (Fig. 3-c, $\alpha = 8^{\circ}$ and 12°) are similar to those for the smaller incidence angle, but with denser distributions on the flank of the body, and the contours are curved back towards the leeward centerline in the rear part. As shown in the $C_{p\bar{\tau}}$ contours (Fig. 3-d, $\alpha = 8^{\circ}$ and 12°), the larger magnitude of streamwise pressure gradients than those for $\alpha = 4^{\circ}$ are observed, whose values mostly show favorable pressure gradients in the windward and forward side, and the reverse holds true in the leeward and rear side. The boundary layers influenced by the pressure gradients finally result in the open separation as mentioned above. As long as global observation is concerned, the separation patterns for $\alpha = 8^{\circ}$ and 12° are essentially different from that for $\alpha = 4^{\circ}$, i.e., the open-type separations are observed. Figs. 5-a and -b show global separation patterns of the observation by Nishio et al.¹⁰) for $\alpha = 11^{\circ}$ and the present solution for $\alpha = 12^{\circ}$, respectively. Nishio et al.¹⁰ classified lines (A) and (B) as the primary and secondary separation lines, respectively, and (C) the reattachment line. Similar features of the separation pattern are observed in the present results, i.e., lines (i), (ii), (iii) in Fig. 5-b can be related to (A), (B), (C) in Fig. 5-a.



Fig. 5 Global view of separation patterns: (a) $\alpha = 11^{\circ}$ [Nishio et al.(1991)]; and (b) $\alpha = 12^{\circ}$ (present results).

3.2 Local observation

For $\alpha = 0^{\circ}$, as seen in the local frictional streamlines (Fig. 6-a, $\alpha = 0^{\circ}$) and the normalized shear stress vectors (Fig. 6-b, $\alpha = 0^{\circ}$), the directions of shear stresses are altered several times in the postseparation region, i.e., several girthwise vortices exist in the separation bubble. This argument is further supported by the streamlines on crossflow planes shown in Fig. 10 for $\alpha = 0^{\circ}$. The streamlines in the figure are obtained based on the crossflow vector fields in the Z=0 plane, therefore its physical meaning may be misleading, but will provide useful information to understand the flow structure. Four girthwise vortices with same rotational directions and one vortex with different rotational direction are observed in the figure. Those formations of girthwise vortices are analogous to the Stokes layer, which is typically observed in the laminar flow solutions. This may be worthy noting that no other experimental or computational studies related to the present problem have reported it as far as the authors' literature survey is concerned. The iso- $|\mathbf{L}_{\mathbf{V}}^*|$ surfaces (Fig. 5, $\alpha = 0^\circ$) display several stripes in the post-separation region, which correlates the above mentioned. Note that the normalized shear stress vectors indicate the location of separation line at X = 0.356.

For $\alpha = 4^{\circ}$, the local frictional streamlines (Fig. 6-a,



Fig. 6 Local comparison of solutions: (a) frictional streamlines; and (b) normalized shear stress vectors ; rows, $\alpha = 0^{\circ}$, $\alpha = 4^{\circ}$, $\alpha = 8^{\circ}$, and $\alpha = 12^{\circ}$, respectively.

 $\alpha = 4^{\circ}$) and the normalized shear stress vectors (Fig. 6-b, $\alpha = 4^{\circ}$) show that the location of separation line agrees well with the observation of Nishio et al.¹⁰) (Fig. 8-a), which are nearly identical. The present results indicate that the flow reversal points on the windward and leeward centerlines are saddle points in the shear stress vector field, which are located at X=0.398 and 0.464, respectively. Fig. 7 shows the zero shear stress contours, i.e., contours such that each component of shear stress vector is zero. The figure clearly identify the locations of singular points for this incidence angle. The local separation pattern of the present solution is also shown in Fig. 9. A focal node is observed near the windward centerline in the present results, which is also indicated by computational results of Kim et al.²⁶⁾²⁷⁾ for the similar α . On the other hand, Nishio et al.¹⁰⁾ observed no focal node in the same α . Note that the focal node is located very close to the saddle point on the windward centerline,

therefore it would be very difficult to detect the point by the flow visualizations, even though the point really exists.

In fact, most of computational or experimental studies related to the present problem yield different arguments regarding the location and type of singular points for this incidence angle. Nishio et al.¹⁰) and Kim et al.²⁶⁾²⁷⁾ observed a saddle point on the flank of the body (Fig. 8-a); however, it is not clear in the present results shown in Fig. 9. The singular point on the leeward centerline is the saddle point in the present solution or Kim et al.²⁶⁾²⁷⁾; however, it is the nodal point in the observation of Nishio et al.¹⁰). On the other hand, Fig. 10 for $\alpha = 4^{\circ}$ display the similar configuration of girthwise vortices as that descried for $\alpha = 0^{\circ}$. In the figure, shaded area corresponds to the region with negative vertical velocity (W). Those girthwise vortices have same rotational directions. Note that no girthwise vortex is observed in the crossflow sections for $\alpha = 12^{\circ}$ (Fig. 10).

For $\alpha = 8^{\circ}$ and 12°, the global observation identified the open-type separation. More details of the flow structures in the post-separation region are displayed in the local frictional streamlines (Fig. 6-a, $\alpha = 8^{\circ}$ and 12°) and the normalized shear stress vectors (Fig. 6b, $\alpha = 8^{\circ}$ and 12°). For $\alpha = 8^{\circ}$, the normalized shear stress vectors indicate the saddle points on the centerlines at X=0.418 and 0.467 at the windward and leeward sides, respectively, and those for $\alpha = 12^{\circ}$ are at X=0.440 and 0.468, respectively. The local separation pattern observed by Nishio et al.¹⁰⁾ for $\alpha = 11^{\circ}$ and that of the present results for $\alpha = 12^{\circ}$ are shown in Figs. 8 and 9, respectively. The notations of separation lines (i), (ii), and (iii) in Fig. 9 are same as those in Fig. 5. For $\alpha = 8^{\circ}$, the separation pattern displayed in Fig. 6-a indicates the closed-type separation near the tail, i.e., the separation pattern can be classified as a combination of the open- and closed-type separations.

The separation pattern for $\alpha = 12^{\circ}$ shown in Fig. 6-a is somehow similar to that for $\alpha = 8^{\circ}$, i.e., there might be bubble formation near the tail, but more details displayed in Fig. 9 indicate that the separation pattern can not be classified by the conventional separation models, i.e., the open- or closed-type, or combination of the both types. As shown in Fig. 8, Nishio et al.¹⁰⁾ conjectured the separation pattern for $\alpha = 11^{\circ}$ as a combination of open and closed sepa-



Fig. 7 ZERO shear stress contours for $\alpha = 4^{\circ}$.



Fig. 8 Local view of separation patterns [Nishio et al. (1991)]: (a) $\alpha = 4^{\circ}$; and (b) $\alpha = 11^{\circ}$.

rations, and defined the saddle point on the flank of the body as well as the separation bubble denoted by dashed line. The argument of Nishio et al. ¹⁰⁾ might be reasonable under concept of the open and closed separations, which is often called the Maskell-Wang model. In the present results, the reversed flow region can not clearly be identified as a separation bubble model of Maskell and Wang, although a saddle point (S) exists on the flank of the body, more specifically, at the end of the reattachment line denoted as (iii) in Fig. 9.

3.3 Observation in near wake region

In this section, discussions are mainly focused on the flow structures in near wake region for $\alpha = 4^{\circ}$ and 12°, where the typical closed- and open-type separations are involved, respectively. Fig. 11 shows the streamlines on crossflow planes for $\alpha = 4^{\circ}$ at X = 0.46through 0.80. The streamlines are obtained in the similar manner described for Fig. 10. Note that the darker shaded area corresponds to the region with negative axial velocity (U). The figures clearly show the generation and development of a vortex in the postseparation and near wake, and its evolution into a longitudinal vortex in the wake. It is seen that the longitudinal vortex is continued into the separation bubble around X=0.54, and goes upstream in the reversed flow region. The vortex core is located near the bubble boundary, and approaches very close to the bubble boundary around X=0.46. Unfortunately our analysis could not locate the final destination of the vortex, but it may be near the separation line denoted as (L) in Fig. 9. It may also be possible that the vortex is connected to the girthwise vortex which was discussed earlier.

On the other hand, three vortices are observed for $\alpha = 12^{\circ}$, which are denoted as (a), (b), and (c) in Fig. 12. The reversed flow region (darker shaded area) is clearly observed in the figure, which supports the earlier discussion stated that the separation pattern for this incidence angle is a combination of the open- and closed-type separations, but without clear separation bubble defined in the Maskell-Wang model. Vortex (a) is supposedly generated along the primary separation line [denoted as line (i) in Fig. 5 or 9] further upstream. Although the formation of the vortex is not as clear as others around X=0.44, its strength increases as X increases in the range of X shown in the



Fig. 9 Localview of separation patterns (present results): (a) $\alpha = 4^{\circ}$; and (b) $\alpha = 12^{\circ}$.



Fig. 10 Streamlines on crossflow planes (Z=0 slices).



Fig. 11 Streamlines on crossflow planes for $\alpha = 4^{\circ}$ (X=const.slices).

figure. Vortex (c) has the same rotational direction as vortex (a), and vortex (b) the different rotational direction as others. The cores of vortices (b) and (c) are located right outside the separation bubble. It is important to note that the formations of vortices (b) and (c) are similar to that for $\alpha = 4^{\circ}$, i.e., the vortex cores are observed very close to the bubble boundary around X=0.46; but those are located outside the bubble for $\alpha = 12^{\circ}$. As is the case for $\alpha = 4^{\circ}$, the origins of vortices (b) and (c) could be detected by our analysis. Those are likely along the separation lines (ii) and (i) in Fig. 9 for vortices (b) and (c), respectively.

Fig. 13 shows the iso-value surfaces of $\nabla^2 p$ for $\alpha = 4^\circ$. Tanaka and Kida³²⁾ noted that $\nabla^2 p$ can be used to describe the vortex structure in the shear flow. In the work, $\nabla^2 p$ is divided into two parts: $\nabla^2 p = 0.5 w_i w_i - S_{ij} S_{ji} = Q \cdot D$, i.e., vorticity (Q) and rate of strain (D) parts. Iso- $\nabla^2 p$ surfaces will display vortex-tube like structure if Q > D, and vortex-sheet like structure if $Q \approx D$. The iso- $\nabla^2 p$ surfaces for Q - D > 100, Q > 200 in Fig. 13 are more likely vortex tube, whose details in near wake region are discussed above. The figures also indicate girthwise vortex formations in the separation bubble, which correspond to the discussion regarding Fig. 10. Similar analysis for sphere was made by Yamada and Miyata³³.



Fig. 12 Streamlines on crossflow planes for $\alpha = 12^{\circ}$ (X=const.slices).

3.4 Lifting and drag forces

For engineering purpose, the resultant hydrodynamic or aerodynamic forces acting on the body is of great interest in conjunction with the flow patterns. Integration of the pressure and shear stress distributions over the body surface yields the pressure and frictional forces, respectively. Fig. 14 shows the comparison of lift and drag coefficients based on the surface area (Cl and Cd, respectively) for all incidence angles considered in the present computation. In the figure, the frictional and pressure parts of the Cl or Cd are denoted as CF and CP, respectively. The Clrapidly increases as α increases, where the contribution of frictional part is very small, which indicates the maximum value at $\alpha = 6^{\circ}$, and small negative values at $\alpha > 10^{\circ}$. It is noted that the Cl vs. α show somehow nonlinear characteristics, as seen for the case of the slender bodies or highly swept delta wings with sharp leading edge.

On the other hand, the frictional and pressure parts of the Cd show obvious nonlinear (nearly parabolic) increase as α increases. The nonlinear characteristics of the pressure part is predicted by the lifting line theory for finite span, i.e., the induced drag is proportional to Cl^2 . In the recent computational study for a sailing yacht³¹⁾, the frictional part of Cd is almost equal as α increases, however the present results show quite different characteristics, which may be due to



Fig. 13 Iso- $\nabla^2 p$ surfaces for $\alpha = 4^{\circ}$.



Fig. 14 Lift and drag coefficients.

the changes in separation patterns. It is interesting to note that Cd at $\alpha=0^{\circ}$ is 0.0045, while that of sphere is 0.1175^{34} , which implies that the post-separation area of a prolate spheroid is much smaller than that of a sphere.

4. Concluding remarks

This work concerns a computational study of threedimensional laminar separation on a prolate spheroid The fully-elliptic NS and continuity at incidence. equations are solved using the regular grid, finiteanalytic discretization, a PISO-type velocity-pressure coupling algorithm, and method of lines. An O-Otype body-conforming grid, which is suitable for the detailed study of the present problem, is used in the computation. In the numerical results presented for a 6:1 prolate spheroid, detailed flow structures in the post-separation region and near wake are displayed, such as the generation and development of the girthwise and longitudinal vortices or the separation patterns which can not be classified by the conventional open- and closed-type separation models. The conclusions regarding the present results are summarized as follows:

1) The present numerical approach is capable for the detailed study on three-dimensional laminar boundary layers and wakes involving separation around a prolate spheroid, i.e., the computation can be continued in the post-separation region. However, the unsteady characteristics of the flow may not be reproduced due to the limitation of numerical scheme.

2) Global and local observations of the present results agree well with those of other computational and experimental studies, i.e., the closed-type separation is observed for $\alpha = 0^{\circ}$ and 4° , and combination of open and closed types for $\alpha = 8^{\circ}$ and 12° . Especially for $\alpha = 12^{\circ}$, the reversed flow region can not clearly be identified as a separation bubble of Maskell-Wang model.

3) The features of the frictional streamlines are discussed in conjunction with the development and separation of the boundary layers, which are related to the shear-stress distributions, pressure distributions (pressure pocket), and streamwise pressure gradients. The iso-value surfaces of $|\mathbf{Lv}^*|$ and $\nabla^2 p$ perspectively present the flow structure associated with three-dimensional separation of the boundary layers and formation of the vortices.

4) Streamlines on crossflow planes are used in the present analysis, which clearly indicate the generation and development of the girthwise and longitudinal vortices. The several girthwise vorticies are observed for $\alpha = 0^{\circ}$ and 4° , which are analogous to the Stokes layers. The constant-X slices display the development of the longitudinal vortices. One and three longitudinal vortices in near wake region are observed for $\alpha = 4^{\circ}$ and 12° , respectively, and those formations are different between the angles, i.e., inside and outside the bubble, respectively.

5) Nonlinear characteristics of Cl vs. α as well as Cd vs. α are observed. Contribution of frictional part in Cl is very small. Frictional part of Cd also show clear nonlinear increase as a increases, which may be due to the change in separation patters. The quantitative evaluation of the results are not possible, because the present solutions are for laminar case.

This study presents a laminar solution, thus the discussion of the results is limited to qualitative analysis, since quantitative experiments in the laminar flow are relatively few. In general, the natural transition from laminar to turbulent flows is observed as *Re* increases, which results in significantly different flow fields or separation patterns. The present computation can be extended to turbulent flow by introducing a suitable turbulence model. Recent progress on the two-layer k- ϵ turbulence model approach of Tahara³⁰) offers encouragement in extending the techniques for the present application.

Finally, the issues that need to be addressed while further developing and validating the present study are summarized as follows: further analysis on the present results through more detailed comparisons with available experimental data; extension of the present study for the turbulent transition and separation problem; and modification of the numerical method regarding the computational grid and the solution algorithm for higher accuracy and efficiency. Also it is of interest to extend the present numerical approach to the time-accurate scheme, in order to investigate the unsteady characteristics of the flow field associated with the three-dimensional flow separation and vortex shedding.

References

- Maskell, E.C., "Flow separation in Three-Dimensionals," RAE Report Aero. 2565, 1955.
- Legendre, R., "Separation de L'ecoulement Laminaire Tridimensio-nell," Rech. Aero., No. 54, 1956.
- Lighthill, M.J., "Attachment and Separation in Three-Dimensional Flow," Laminar Boundary Layers, ed. L. Rosenhead, Oxford Univ. Press, 1963, pp. 72-82.
- Wang. K. C., "Separation Patterns of Boundary Layer over an Inclined Body of Revolution," AIAA Journal, Vol. 10, 1972, pp. 1044-1050.
- 5) Tobak, M. and Peake, D.J., "Topology of Three-Dimensional Separated Flows," Ann. Review Fluid Mechanics, Vol. 14, 1982, pp. 61-85.
- Han, T. and Patel, V.C., "Flow Separation on a Spheroid at Incidence," J. Fluid Mechanics, Vol. 92, 1979, pp. 643-658
- Werle, H., "Principaux Types de Decollement Libre Observes sur Maquettes Ellipsoidales," ON-ERA, France, Report FR ISSN0078-3781,1985.
- Costis, C.E., Hoang, N.T. and Telionis, D.P., "Laminar Separating Flow over a Prolate Spheroid," J. Aircraft, Vol.26, 1989, pp.810-816.

- Nishio, S., Tanaka, I., Noritake, Y., and Nishikawa, T., "Study on Separated Flow around Prolate Spheroid at Incidence," J. Kansai Society of Naval Architects, Japan, No. 207, December 1987, pp. 45-52 [Japanese].
- 10) Nishio, S., Noritake, Y., and Tanaka, I., "Experimental Study of Surface Pressure Distribution on a Prolate Spheroid at Small Incidence," J. Kansai Society of Naval Architects, Japan, No. 211, March 1991, pp. 47-54 [Japanese].
- Wang, K.C., Zhou, H.C., Hu, C.H. and Harringyon, S., "Three-Dimensional Separated Flow Structure over Prolate Spheroids," Proc. Roy. Soc., London, Vol. A421, 1990, pp. 73-90.
- 12) Wang, K.C. "Boundary Layer over a Blunt Body at High Incidence with an Open-type of Separation," Proc. Roy. Soc., London, Vol. A340, 1974, pp. 33-55.
- Wang, K.C. "Boundary Layer over a Blunt Body at Extremely High Incidence," Phys. Fluids, Vol. 17, 1974, pp. 1381-1385.
- 14) Wang, K.C. "Boundary Layer over a Blunt Body at Low Incidence with Circumferential Reversed Flow," J. Fluid mechanics, Vol. 72, 1975, pp. 49-65.
- 15) Hayashita, S. and Himeno, Y., "On Limiting Streamlines Expressing Three-Dimensional Separation," J. Society of Naval Architects of Japan, Vol. 143, 1978, pp. 17-25 [Japanese].
- 16) Patel, V.C. and Choi, D.H., "Calculation of Three-Dimensional Laminar and Turbulent Boundary Layers on Bodies of Revolution at Incidence," Turbulent Shear Flows II, Springer-Verlag, 1981, pp. 199-217.
- Cebeci, T., Khattab, A.K. and Stewartson, K., "Three-Dimensional Laminar Boundary Layers and the ok of Accessibility," J. Fluid mechanics, Vol. 107, 1981, pp. 57-87.
- 18) Cebeci, T. and Su, W., "Separation of Three-Dimensional Laminar Boundary Layers on a Prolate Spheroid," J. Fluid mechanics, Vol. 191, 1988, pp. 47-77.
- 19) Patel, V.C. and Baek, J.H., "Boundary Layers and Separation on a Spheroid at Incidence," AIAA Journal, Vol. 23, 1985, pp. 55-63.
- 20) Pan, D. and Pullian, T.H., "The Computation of Steady 3-D Separated Flows over Aerodynamic

Bodies at Incidence and Yaw," AIAA-86-0109, 1986.

- 21) Rosenfeld, M., Israeli, M. and Wolfshtein, M.,
 "Numerical Study of the Skin Friction on a Spheroid at Incidence," AIAA Journal, Vol. 26, 1988, pp. 129-136.
- 22) Vasta, V.N., Thomas, J.L. and Wedan, B.W., "Navier-Stokes Computations of a Prolate Spheroid at Angle of Attack," J. Aircraft, Vol.26, 1989, pp.986-993.
- 23) Queutey, P., "Resolution des equations de Navier-Stokes tridimensionnelles. Application au calcul sur des corps en incidence," Doctoral Thesis, ENSM, Uni. Nantes, France, 1989.
- 24) Deng, G., Piquet, J. and Queutey, P., "Navier-Stokes Computations of Vortical Flows," AIAA-90-1628, 1990.
- 25) Nishio, S., "Computation of Flow around Spheroid at Small Incidence and Study on Vorticity Field," J. Society of Naval Architects of Japan, Vol. 168, 1990, pp. 9-20 [Japanese].
- 26) Kim, S.E. and Patel, V.C., "Laminar Flow Separation on a Spheroid at Incidence," Proc. AIAA 22nd Fluid Dynamics, Plasma Dynamics & Lasers Conference, Honolulu, Hawaii, 1991.
- 27) Kim, S.E., "Numerical Studies of Three-Dimensional Flow Separation," Ph.D. Thesis, The University of Iowa, Iowa City, IA, 1991 (unpublished).
- 28) Tahara, Y., "Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments," J. Kansai Society of Naval Architects, Japan, No. 220, September 1993, pp. 29-47.
- 29) Tahara Y. and Stern, F., "A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number," Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994.
- 30) Tahara, Y., "An Application of Two-Layer k- ϵ Model to Ship Flow Computation," J. Society of Naval Architects of Japan, Vol. 177, 1995, pp. 161-176.
- 31) Tahara, Y., "Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht - For a Canoe Body Case," J. Kansai Society of Naval Architects, Japan, No. 224, September 1995, pp. 1-11.

- 32) Tanaka M. and Kida S., "Characterization of vortex tubes and sheets," J. Physics of Fluids A, Vol. 5, No. 9, September 1993, pp. 2079-2081.
- 33) Yamada, Y., and Miyata, H. "Computational Study of Large Eddy Structure of Flows past Bluff Body and Oceanic Topography," J. Society of Naval Architects of Japan, Vol. 173, 1993, pp. 19-26.
- 34) White, F.M., "Fluid Mechanics," McGraw-Hill Book Company, 1986.

Discussion

[Discussion] (横浜国立大学)池畑 光尚

大変奇麗な流れ模様を見せて頂き、感心致しました. でも疑問に感じた事についてお尋ね致します.物体表 面近くの剪断流中の流速の極く遅い流れを計算してい るのにどうしてこんなに滑らかに流線が繋がって出て くるのでしょう.特に特異点の所では流線は分岐した り、渦巻いたりするので、計算が停まってしまったり、 不安定になったり、ぐるぐる回りに陥ち込んだりする 事はないのでしょうか.

[Author's Reply]

流線追跡計算は物体の極近傍では確かに不安定にな りがちです.流線が物体に向かって進行する場合、当 然ながら計算は壁面の直前でストップします。物体表 面の特異点から出てくる流線を求める場合は逆追跡と いう手法を用いています。すなわち、流れの下流側か ら上流へ向けて、特異点の直前まで逆方向に流線追跡 を行う訳です。

A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht *1

By Yusuke TAHARA (Member) *2

A multi-domain method is set forth for calculating boundary-layer and wake flows around the International America's Cup Class (IACC) sailing yacht. The physical domain is divided into two parts, i.e., the port- and starboard-side domains, where those centerplane (matching) boundaries include keel and rudder surfaces. The Reynolds-averaged Navier-Stokes and continuity equations are solved with the Baldwin-Lomax turbulence model, using a body conforming grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. An overview is given of the present approach, and numerical results are presented and discussed for an IACC yacht for zero and nonzero yaw angles, including the comparisons with the canoe body solutions in order to evaluate influences of keel and rudder on boundary-layer flow near the hull surface. The present approach appears to be capable for the detailed study of the flow fields around the entire boat as well as the drag and lifting forces acting on it. Lastly, some concluding remarks are made concerning requirements for future work of the present study.

Keywords: Multi-domain Method, Navier-Stokes Equations, America's Cup Sailing Yacht, Finite-analytic Scheme

1. Introduction

Accurate prediction of hydrodynamic forces acting on hull and appendages is extremely important in IACC (International America's Cup Class) sailing yacht design. Boat designers have demanded for reliable numerical tools which can be used in conjunction with experimental approaches. Although quite a few numerical methods have been developed for the IACC $boat^{1)-7}$, relatively few are based on the Reynoldsaveraged Navier-Stokes (RaNS) equation method. In order to correctly estimate viscous resistance, RaNS equations must be solved to capture complex flow aspects around IACC boat, in which three-dimensional separation and vortex shedding from appendages are usually involved. Tahara¹⁾ and Miyata⁶⁾⁷⁾ developed numerical methods based on RaNS equations for IACC boat, but appendages like keel and rudder are not considered. When the boat advances with nonzero yaw angle, appendages generally produce major part

of lifting force and associated induce drag, therefore, must be included in the viscous-flow computation to account for them.

The present work demonstrates for the first time the feasibility of RaNS equation method for IACC sailing yacht including keel and rudder. A numerical method is developed, which is based on extension and modification of the author's previous work¹) by the use of a multi-domain method. The physical domain is divided into two parts, i.e., the port- and starboard-side domains, where those centerplane (matching) boundaries include the keel and rudder surfaces. The RaNS and continuity equations are solved with the Baldwin-Lomax turbulence model, where the free surface is considered to be flat and treated as a plane of symmetry. The present approach enables the detailed study of the flow fields around the entire boat as well as the drag and lifting forces acting on it, which are especially important for the case with yaw angle. An overview is given of the present approach, and numerical results are presented and discussed for an IACC yacht for zero and nonzero yaw angles, including the comparisons with the canoe body solutions¹⁾ in order to evaluate influences of keel and rudder on boundary-layer flow

^{*1} Read at the Spring Meeting of Kansai Society of Naval Architects, Japan, May 22, 1996, Received June 10, 1996

^{*2} Osaka Prefecture University

near the hull surface. Lastly, some concluding remarks are made concerning requirements for future work of the present study.



Fig. 1 Definition sketch of coordinate system, solution domain and boundaries.



Fig. 2 Computational grid: (a) body crossplane (X=0.5); (b) side view; and (c) hull surface.

2. Computational Method

The present numerical approach is based on extension of that used in the previous work¹⁾, with some modifications for the multi-domain method. The unsteady RaNS and continuity equations with the Baldwin-Lomax turbulence model are solved for the mean-velocity components (U, V, W), pressure p, and eddy viscosity ν_t . The transport equations for (U, V, W) are written in Cartesian coordinates (X, Y, Z) in the physical domain and partially transformed into numerically generated boundary-fitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) . Variables are non-dimensionalized using the freestream velocity U_0 (=1), ship length L (=1), and density ρ . The transformed equations are solved using a regular grid, finite-analytic discretization, a PISO-type velocity-pressure coupling algorithm, and the method of lines. Although the present solutions are for steady flow, the equations are solved in unsteady form with time serving as a convergence parameter. In use of the Baldwin-Lomax turbulence model, the length scales are given by the distance from the nearest body or appendage surface.

Consider a ship fixed in the uniform onset flow $U_0 = (U_0 \cos \alpha, U_0 \sin \alpha, 0)$ as depicted in Fig.1 (where α is yaw angle). Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward. The solution domain is also shown in the figure. The physical domain is divided into two parts, i.e., the port- and starboard-side domains, where those centerplane (or matching) boundaries include the keel and rudder surfaces.

As mentioned earlier, the present numerical approach utilizes the multi-domain method. Although, in principle, it is possible to establish a correspondence between any physical domain and a single rectangular computational block for general three-dimensional configurations, the resultant computational grid is likely to be too skewed and irregular to be usable when the boundary geometry is complicated. There are two solutions to the problem: (1) the use of unstructured grid system and associated solution method (e.g., $Hino^{8}$); and (2) the use of multi-domain method (e.g., Hirata⁹⁾). In the multi-domain method, the physical domain is segmented into contiguous subdomains, each bounded by six curved sides and each of which transforms to a rectangular block in the computational domain, with a grid generated within each sub-domain.

This then allows both the grid generation and numerical solutions on the grid to be constructed to operate in a rectangular computational domain, regardless of the shape or complexity of the full physical domain. The full domain is treated by performing the solution operation in all of the rectangular computa-



Fig. 3 Comparison of solutions for $\alpha = 0^{\circ}$: (a) axial-velocity contours; (b) crossplane-velocity vectors; and (c) C_p contours; rows, X=0 (FP), X=0.25, X=0.50, X=0.75, X=1.0 (AP), and X=1.1, respectively.



Fig. 4 Comparison of solutions for $\alpha = 10^{\circ}$: (a) axial-velocity contours; (b) crossplane-velocity vectors; and (c) C_p contours; rows, X=0 (FP), X=0.25, X=0.50, X=0.75, X=1.0 (AP), and X=1.1, respectively.



g. 5 Grobal comparison of solutions on hum surface for $\alpha = 0^{\circ}$: (a) C_p contours; (b) frictional streamlines; and (c) C_{τ} contours.

tional domains. Partial differential equation solution procedures written to operate on rectangular domains can be incorporate into a code for general configurations in a straightforward manner, since the code only needs to treat a rectangular block. The entire physical domain then can be treated in a loop overall the blocks.

The general curved surfaces bounding the subdomains in the physical domain form internal interfaces across which information must be transferred, i.e., from the sides of one rectangular computational block to those of another. These interfaces occur on the same, or another, block, since both correspond to the same physical surface. In the present application, only two sub-domains are used, and grid lines on the interfaces are completely continuous. It is relatively easy to increase the number of sub-domains in future work, so that more appendages, e.g., the winglet-bulb, can be included in the present hull-rudder and -keel configuration.

In terms of the notation of Fig.1, the boundary conditions on each of the boundaries are described as follows: on inlet plane Si and outer boundary So, the boundary conditions are provided on the basis of the freestream values, i.e., $U=U_0 \cos \alpha$, $V=U_0 \sin \alpha$, W=0and p is linearly extrapolated from the interior nodes;



Fig. 6 Global comparison of solutions on hull surface for $\alpha=5^{\circ}$: (a) C_p contours; (b) frictional streamlines; and (c) C_{τ} contours.

on exit plane Se, axial diffusion is negligible so that the exit conditions used are $\partial^2(U, V, W)/\partial X^2 = \partial p/\partial X = 0$; on waterplane Sw, wave effects are not considered in the present study, so that the conditions imposed are $\partial(U, V, p)/\partial Z = W = 0$; and on the hull, keel, and rudder surfaces, i.e., Sh, Sk, and Sr, boundary conditions used are $U=V=W=\partial p/\partial n=0$ (where *n* is normal to the surface). The conditions of the matching boundaries Sc are given by the interpolation using the values at the adjacent interior nodes of the two domains, and updated at every computational time step. In this study, the initial conditions are taken from the freestream values.

3. Results and Discussion

In the present study, the computational results are presented for the canoe $body^{1}$ with the dummy keel and rudder, which is due to the limitation under the non-disclosure agreement. Therefore, real keel and rudder shapes and their locations are somewhat different from those used in the present study. In this computation, the locations of keel center X_{ck} , maximum thickness T_k , cord length L_k and maximum depth D_k are $(X_{ck}=0.5,$ $T_k=0.001, L_k=0.06, D_k=0.2)$, and those for rudder, i.e., X_{cr} , T_r , L_r , and D_r are $(X_{cr}=0.985,$ $T_r=0.0005$, $L_r=0.03$, $D_r=0.15$). The present keel and rudder surfaces $Y=Y_k(X,Z)$ and $Y=Y_r(X,Z)$ are given by the parabolic equations as follows: $Y_k(X,Z) = 4T_k(Z/D_k+1)^2((X-X_{ck})/L_k+.5)((X_{ck}-X)/L_k+.5);$ and $Y_r(X,Z) = 4T_r(Z/D_r+1)^2((X-X_{cr})/L_r+.5),$ respectively.



Fig. 7 Global comparison of solutions on hull surface for $\alpha = 10^{\circ}$: (a) C_p contours; (b) frictional streamlines; and (c) C_{τ} contours.

The computational grids were obtained using the elliptic-algebraic method of Tahara¹⁾. The inlet, exit and outer boundaries are placed at X=(-0.4, 2.0) and r=1, respectively, where r is the radial direction as shown in Fig.1. The first grid point off the body or appendage surface is located in the viscous sublayer, i.e., $y^+ = RnU_{\tau}y = Rn(\tau_w/.5\rho U_0^2)^{0.5}y < 2$, where y is the distance from the wall. The numbers of grids used are $125 \times 50 \times 39$ (=243,750) in the axial, radial, and circumferential directions, respectively, where the combination and grid concentration near the keel and rudder tips are found to be optimum in the numerical tests as well as other information regarding grid study with similar numerical scheme¹⁾¹⁰⁾¹¹⁾. Figs.2-a through -c show partial views of the present computational grid for the typical body crossplane and body surface. The conditions for the computation were as follows: L=1; $U_0=1$; Rn=4,500,000. The workstation (DEC-AlphaStation200/4/233) time and central memory were about 63 hours and 128MB for each incidence angle. The convergence criterion was that the change in solution be less than about 0.05% for all variables, which was satisfied in 1000 global iterations.

Figs.3 and 4 show the axial-velocity (U) contours, crossplane-velocity vectors (V-W), and pressure $(C_p = p/.5)$ contours for several representative sections. Figs.5 through 7 provide global comparisons of the hull-surface-pressure contours, the frictional (or limiting) streamlines, and the wall-shear-stress $(C_{\tau}=\tau_w/.5\rho U_0^2)$ contours for $\alpha=0^\circ$, 5°, and 10°, respectively. Local comparisons of the solutions on the hull surface are provided for regions near the hull-keel and -rudder junctions in Figs.8 and 9, respectively. The keel- and rudder-surface-pressure distributions and the frictional streamlines are shown in Figs.10 and 11, respectively. The view areas for those local comparisons are specified in Fig.2. Fig.12 shows the isovalue surfaces of the magnitude of modified Lamb vector $|\mathbf{L}_{\mathbf{v}}^*| = |\mathbf{L}_{\mathbf{v}}|/|\mathbf{U}|$ [where $\mathbf{L}_{\mathbf{v}} = \omega \times \mathbf{U}, \mathbf{U} = (U, V, W),$ and ω is the vorticity vector] and modified Lamb scalar (helicity) $L_s^* = L_s / |U|$ (where $L_s = \omega \cdot U$) for $\alpha = 10^\circ$. Lastly, Fig.13 compares the lift and drag coefficients for all α considered in the present computation. In the following, computational results are presented and discussed, in which variables are non-dimensionalized using the ship length L, the free-stream speed U_0 and fluid density ρ . Note that the rudder trailing edge is located at AP, and X=0 and 1.0 correspond to FP and AP, respectively. Reader may refer to Tahara¹⁾ for the canoe-body solutions, which will be compared with the present results in the following discussion.

3.1 Comparison of Results at Cross Sections for $\alpha = 0^{\circ}$

At X=0 and 0.25 (Fig.3), the U and C_p contours and the V-W vectors display small differences between the present and previous canoe-body results¹⁾. The differences are clear in the afterbody. At X=0.5 and 0.75 (Fig.3), influences of the keel on the flow fields are clearly observed. At X=0.5, the location of keel center, the U contours show continuous growth of the boundary layer near the hull, which is thicker near the hull-keel junction than the waterline, and development of the thin boundary layer near the keel is also seen. The C_p contours at X=0.5 are completely different from those of the canoe-body case¹⁾, where the lowest value is observed near the keel and slightly below the hull-keel junction, which is influenced by the



Fig. 8 Local comparison of solutions on hull surface (near hull-keel junction): (a) C_p contours; (b) shear-stress vectors; and (c) frictional streamlines; rows, $\alpha=0^\circ$, $\alpha=5^\circ$, and $\alpha=10^\circ$, respectively.

low pressure on the keel surface. At X=0.75, the U contours display the thick boundary layer characteristics near the hull, and elongate keel wake near the centerplane, where the region of U > 1 is observed right below the keel tip (Z=-0.2). The C_p contours at X=0.75 show negative values everywhere, and similar extents as those of the canoe-body solutions¹), where the lowest value is observed near the centerline.

At X=1 and 1.1 (Fig.3), the U and C_p contours display merging of the boundary layer into the wake and its initial evolution, where clear differences are observed between the present and canoe-body¹⁾ results due to the influences of appendages. At X=1, the location of rudder trailing edge, the U contours indicate region of U < 1 around Z=-0.2, which is deeper than the location of rudder tip (Z=-0.15). This indicates that the flow field at this section is clearly influenced by the keel wake. The C_p contours at X=1show the highest value near the hull-rudder junction. At X=1.1, the U contours still indicate elongate wake near the centerplane.

3.2 Comparison of Results at Cross Sections for $\alpha = 10^{\circ}$

As is the case for $\alpha=0^{\circ}$, influences of appendages are clear in the afterbody (Fig.4). At X=0.5, the C_p contours show significantly high and low values in the port and starboard sides, i.e., the pressure and suctions sides of the keel, respectively. The differences of the pressure between the two sides produce considerably high lifting forces as described later. In conjunction with the pressure fields at the section, the present velocity fields are completely different from those of the canoe-body results¹). The V-W vectors at X=0.5 display the generation of vortex near the keel tip, where the vectors with considerably large magnitude are observed. The U contours at X=0.75 show that the boundary layer is thicker near the starboardside waterline than the port-side waterline, which is


Fig. 9 Local comparison of solutions on hull surface (near hull-rudder junction): (a) C_p contours; (b) shearstress vectors; and (c) frictional streamlines; rows, $\alpha=0^{\circ}$, $\alpha=5^{\circ}$, and $\alpha=10^{\circ}$, respectively.

due to the convergence of the flow near the starboardside waterline as shown in the V-W vectors. The keel wake is also observed in the U contours, which is slightly shifted rightward. The values of C_p contours at X=0.75 are mostly negative, and the deformation of contours are observed, where the V-W vectors indicate remnant of the keel-tip vortex.

The C_p contours at X=1 indicate higher values in the port side than the starboard side, which is analogous to the pressure field near the keel, i.e., those sides correspond to the pressure and suction sides of the rudder, respectively. The V-W vectors at X=1are mostly rightward except in the region very close to the rudder, where the generation of vortex is observed near the tip of rudder. As also seen near the keel tip, considerably large magnitude of vectors are observed. The U contours at X=1.1 show slower recovery in the starboard side than the port side, where the extents of the contours are completely different from those of the canoe-body solutions¹⁾. The V-W vectors at X=1.1 indicate remnant of the rudder-tip vortex. As is the case for X=0.75, the C_p contours at X=1.1 show the deformation of contours near the center of the rudder-tip vortex.

3.3 Comparison of Results on Hull Surface

For $\alpha=0^{\circ}$, the hull-surface C_p and C_{τ} contours (Figs.5-a and -b) are mostly similar to those of the canoe-body results¹; however, some differences are also observed in the region near the keel and rudder (Figs.8-a and 9-a), and near the centerline especially for C_{τ} . As shown in Figs.5-a and 8-a, the lowest value of C_p contours is seen in the region very close to the keel and slightly after the keel center, which is associated with the low pressure distributions on the keel surface near the keel center (Fig.10). Note that the centers of keel and rudder correspond to the locations of their maximum thicknesses. The differences of C_{τ} contours between the present and canoe-body results¹) are highlighted in the afterbody of the hull and near the centerline (Fig.5-c), i.e., values of the present results are somewhat lower, where the shear flows are influenced by the keel wake.

For $\alpha = 5^{\circ}$ and 10° (Figs.6 and 7), influences of keel and rudder are obvious. Near the keel and rudder, considerably high and low pressure regions are located in the port and starboard sides, respectively (Fig.8-a for $\alpha = 5^{\circ}$ and 10°), and those are higher and lower for $\alpha = 10^{\circ}$, respectively. The shear-stress vectors for $\alpha = 5^{\circ}$ and 10° (Fig.8-b) indicate considerably large magnitude of the vectors near the keel leading edge, and similar aspects are observed near the rudder leading edge (Fig.9-b). For $\alpha = 10^{\circ}$, the frictional streamlines near the keel (Fig.8-c) indicate a focal point near the trailing edge in the starboard side. Similar singularities may exist near the hull-rudder junction (Fig.9-c for $\alpha=5^{\circ}$ and 10°); however, the clear definition was not possible due to insufficient resolution of the present computational grid. The values of C_{τ} for $\alpha = 5^{\circ}$ and 10° are generally higher in the port side than the starboard side, which correlates the thicker boundary-layer in the starboard side, and somewhat lower values are observed in the afterbody where the shear flows are influenced by the keel wake.

3.4 Comparison of Solutions on Keel and Rudder Surfaces

Fig.10 provides the C_p contours and frictional streamlines on the keel surface. Note that the mainstream direction is rightward, i.e., positive X direction in the figures. The frictional streamlines for $\alpha=0^{\circ}$ are nearly parallel to the bottomline of the hull. For $\alpha=5^{\circ}$ and 10°, frictional streamlines near the tip are directed downward in the port side, which is caused by the lower pressure region very close to the tip, while those in the starboard side are first directed slightly upward near the leading edge then downward near the trailing edge. The shear-flow structures near the tip are directly related to the generation of the tip vortex, while the stronger tip vortex for $\alpha = 10^{\circ}$ is implied by the larger downward flows in the port side as shown in the frictional streamlines. Results for $\alpha=5^{\circ}$ and 10° also indicate the stalled regions in the starboard side, where the area is larger for $\alpha = 10^{\circ}$. The stalls occur near the root of keel, which is typical separation pattern for a conventional rectangular wing. In usual

boat design, tapered keels are often used in order to avoid the root stall, which significantly decreases and increases the lift and drag, respectively.

Fig.11 shows the comparisons of solutions on the rudder surface. For $\alpha=5^{\circ}$ and 10° , the frictional streamlines near the tip indicate similar aspects associated with generation of tip vortex as those for keel. As compared to the results on the keel surface, those for rudder show some differences especially in the pressure distributions in the upper part and separation pattern displayed in the frictional streamlines. The pressure field on the rudder surface is clearly influenced by the high pressure near the stern end. The shear flows are also influenced by the thick boundary-layer flow on the hull, e.g., the root stall in the starboard side of the rudder occurs only for $\alpha=10^{\circ}$.

3.5 Iso- $|L_V^*|$ and L_S^* Surfaces

Fig.12 shows the iso-value surfaces of the magnitude of modified Lamb vector $|\mathbf{L}\mathbf{v}^*|$ and modified Lamb scalar $\mathbf{L}_{\mathbf{S}}^*$ for $\alpha=10^\circ$. The iso- $|\mathbf{L}\mathbf{v}^*|$ and $\mathbf{L}_{\mathbf{S}}^*$ surfaces (Fig.12-b for $|\mathbf{L}\mathbf{v}^*|=5$, and Fig.12-c for $\mathbf{L}_{\mathbf{S}}^*=5$, respectively) display perspective views of the shearflow structure and generation, development, and decay of the streamwise vortices about the present hull, keel, and rudder configurations (Fig.12-a). Note that $\mathbf{L}_{\mathbf{S}}^*=5$ highlights the streamwise vortices in a counterclockwise direction. The iso- $|\mathbf{L}\mathbf{v}^*|$ surfaces were also used in author's work on the three-dimensional laminar separation on a prolate spheroid at incidence¹².

The present results are characterized by the flow aspects associated with the generation of streamwise vortices and the root stalls occur on the appendages, which are clearly correlated with the iso-Ls* surfaces. The streamwise vortices in a counter-clockwise direction are mainly generated on the port side of the hull, near the hull-keel and -rudder junctions, the stalled regions on keel and rudder, and the keel and rudder tips. Especially, the tip vortices on the keel and rudder are significant as already described for the V-W vectors (Fig.4). It must be noted that the above-mentioned vortex systems generally interact each other, e.g., between the keel- and rudder-tip vortices. In the present figure, the keel-tip vortex seems to decay faster than expected, however, its continuation into the near wake region is seen in the iso-Ls* surfaces of smaller values. The generation of vortices directly results in an



Fig. 10 Pressure contours and frictional streamlines on keel surface: (a) C_p contours; and (b) frictional streamlines; rows, $\alpha = 0^{\circ}$, $\alpha = 5^{\circ}$, and $\alpha = 10^{\circ}$, respectively.



Fig. 11 Pressure contours and frictional streamlines on rudder surface: (a) C_p contours; and (b) frictional streamlines; rows, $\alpha = 0^{\circ}$, $\alpha = 5^{\circ}$, and $\alpha = 10^{\circ}$, respectively.

increase in drag, therefore, must accurately be predicted and controlled in the boat design.



Fig. 12 Comparison of Iso- $|\mathbf{L_V}^*|$ and $-\mathbf{L_S}^*$ surfaces for $\alpha = 10^\circ$: (a) hull, keel and rudder configurations; (b) Iso- $|\mathbf{L_V}^*|$ surfaces; and (c) Iso- $\mathbf{L_S}^*$ surfaces.

3.6 Lifting and Drag Forces

Integration of the pressure and shear stress distributions over the body surface yields the drag and lifting forces, whose coefficients $(C=F/.5\rho S_h U_0^2)$ are shown in Fig.13. Note that all coefficients are based on the wetted surface area of the canoe body S_h , in order to make relative comparison of resultant hydrodynamic forces. The frictional and pressure parts of the drag and lift are denoted as CF and CP in the figure, respectively, and in the following discussion, those are referred to as $(CF_{hull}, CP_{hull}), (CF_{keel}, CP_{keel})$, and $(CF_{rudder}, CP_{rudder})$, with respect to contributions of the hull, keel, and rudder, respectively.

First, the lift components are considered. In general, the CP part of the lift linearly increases as α increases. The contributions of CF are not significant, in which negative small values are indicated. As is expected, appendages produce major part of the total lifting force. The present CP_{hull} shows larger values than that of the canoe body¹, which is correlated with the larger pressure differences between the port and starboard sides, which are clearly due to the influences of keel and rudder (see Figs.5 through 7). The CP_{keel} show larger values than CP_{rudder} , which is mainly related to the larger surface area of the keel.

Next, the drag components are discussed. For a

canoe-body case¹⁾ (denoted as "Bare" in the figure), the CF_{hull} is almost equal as yaw angle α increases. The CF_{hull} of the present results (denoted as "With append.") slightly decreases as α increases, especially between $\alpha = 5^{\circ}$ and 10° , which can be correlated with the lower C_{τ} distributions on the afterbody influenced by the keel wake (see Fig.7 or Fig.9, $\alpha = 10^{\circ}$). The present CP_{hull} indicates nearly parabolic increase as α increases, as is also shown for the canoe-body case. The CF_{keel} also decreases as α increases, which is related to the separation pattern as discussed earlier (Fig.11), whereas CP_{keel} significantly increases. On the other hand, CF_{rudder} increases between $\alpha=0^{\circ}$ and 5° and decreases between $\alpha=5^{\circ}$ and 10°, while CP_{rudder} generally increases as α increases. These trends of CF_{rudder} and CP_{rudder} can be correlated with the surface flow pattern, especially the root stall observed in the starboard side (Fig.12).

The total pressure part of the drag forces show nearly parabolic increase as α increases, where the contributions of appendages are significant for larger incidence angles. The trends of drag forces could be guessed by the lifting line theory for wings of finite span, i.e., the induced drag is proportional to the square of lifting force. In the present hull-appendage configuration, the appendages generate major part of total lifting force, but also the drags associated with the flow separation and generation of the vortices. This supports one of the key issues in the boat design that appendages must carefully be designed through the consideration of controlling or avoiding them.

4. Concluding Remarks

The present work demonstrates for the first time the feasibility of RaNS equation method for IACC sailing yacht including keel and rudder. A numerical method is developed, which is based on extension and modification of the author's previous work¹⁾ by the use of the multi-domain method. The present approach enables the detailed study of the flow fields around the entire boat as well as the drag and lifting forces acting on it, and evaluation of influences of keel and rudder on boundary-layer flow near the hull surface through the comparison with the canoe body solutions of the previous work¹⁾. In the present numerical results for zero and nonzero yaw angles, details of the flow features near the hull and appendages are displayed, in



Fig. 13 Drag and lift coefficients: (a) drag; and (b) lift, respectively.

which the discussions have been made on the hydrodynamic forces acting on the entire vessel in conjunction with the flow patterns of the frictional streamlines on the hull and appendages. In conclusion, the present results have been shown to be very encouraging; however further validation study must be made through the comparison with available experimental data, as well as the case study of the method to be a design tool for hull or appendage optimization. In fact, those including experimental work are currently in progress in the author's research group, and will be reported in the future publication.

5. Acknowledgments

The author wishes to thank Prof. Y. Himeno at Osaka Prefecture University for his valuable discussions and encouragement.

References

1) Tahara, Y.: Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht - For a Canoe Body Case, J. Kansai Society of Naval Architects, Japan, No. 224, September 1995, pp. 1-11.

- Collected Papers on Technology for Stars & Stripes, Science Applications International Corporation, Annapolis, MD, January 1988.
- Proc. The Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, MD, January 1993.
- 4) Boppe, C.W., Rosen, B.S., Laiosa, J.P. and Chance, B., Jr.: Stars & Stripes '87: Computational Flow Simulations for Hydrodynamic Design, The Eighth Chesapeake Sailing Yacht Symposium, Annapolis, MD., 1987.
- 5) Suzuki, K. and Yoshihara, A.: Computation of Hydrodynamic Forces Acting on Sailing Yacht by Means of Surface Panel Method, J. Society of Naval Architects of Japan, Vol. 178, 1995, pp. 113-124 [Japanese].
- Miyata, H.: Time-Marching CFD Simulation for Moving Boundary Problems, Proc. Twenty-first Symposium on Naval Hydrodynamics, Trondheim, Norway, June 1996.

- Miyata, H.: Dynamic Simulation of Sailing Boat by Finite-Volume Method, Proc. Spring Meeting of Society of Computational Engineering, Vol. 1, May 1996.
- 8) Hino, T., Martinelli, L. and Jameson, A.: A Finite-Volume Method with Unstructured Grid for Free Surface Flow Simulations, Proc. 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, IA, August 1993.
- 9) Hirata, N. and Kodama, Y.: Flow Computation for Three-Dimensional Wing in Ground Effect Using Multi-Bloack Technique, J. Society of Naval Architects of Japan, Vol. 177, 1995, pp. 49-57.
- 10) Stern, F., Zhang, D.H., Chen, B., Kim, H.T. and Jessup, S.: Computation of viscous marine propulsor and wake flow, Proc. 20th ONR Symposium on Naval Hydrodynamics, Santa Barbara, CA, August, 1994.
- 11) Chen, B., Stern, F. and Kim, W.: Zhang, D.H., Kim, H.T. and Jessup, S.: Computation of unsteady viscous marine propulsor and wake flow, Proc. 20th ONR Symposium on Naval Hydrodynamics, Santa Barbara, CA, August, 1994.
- 12) Tahara, Y., Mitarai, S. and Himeno, Y.: A Computational Study of Three-Dimensional Laminar Separation on a Prolate Spheroid at Incidence, J. Kansai Society of Naval Architects, Japan, No. 225, March 1996, pp. 93-105.

Discussion

[Discussion] (船舶技術研究所)児玉 良明 キールなどの3次元翼の揚力値や誘導抵抗値の計算精 度に関連して、翼端渦の解像度をどの程度以上にとれ ばよいか、お考えあるいは御経験をお聞かせください.

[Author's Reply]

斜航しているヨットのキールやラダーの翼端近傍で は圧力勾配が大きく、翼端渦の生成にともなって極め て大きな速度勾配も存在します.従って数値計算を行 う際、どの程度計算格子を集中させるかは重要なポイ ントです.著者の経験では、翼端近傍における格子の radial direction の最小値が 0.001 無次元長以下でない と圧力方程式が発散します.今回の計算における計算 格子の基準は安定性のみを考慮したものでしたが、今 後計算手法の定量的評価を行う際、翼端渦の解像度に ついては再検討すべきであろうと思います.

[Discussion] (船舶技術研究所)日野 孝則

揚力問題を考えるとき、side boundary の位置および境界条件が揚力の計算値に大きく影響すると思いま すが、この点検討されましたでしょうか.

[Author's Reply]

ご指摘の通り、外部境界の位置は流体力の計算値に 大きな影響を与えると思います。著者が2次元および 3次元問題を解いてきた過程で経験的に言えるのは、 1代表長さは確実に物体から離すべきであろうという ことです。あまり離しすぎると今度は数値誤差が介入 してくる恐れもあり、ベンチマークテスト等を行って 計算スキーム固有の特性は確認しておくべきであると 思います。

A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes and Wave Fields for Nonzero Froude Number

Y. TAHARA* AND F. STERN

Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa

Received September 12, 1995; revised March 11, 1996

A large-domain approach is developed for calculating ship boundary layers and wakes and wave fields for nonzero Froude number. The Reynolds-averaged Navier-Stokes and continuity equations are solved with the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. The results are validated through comparisons with data for the Series 60 $C_B = 0.6$ ship model at low and high Froude numbers and results of a precursory interactive approach. Both approaches yield satisfactory results; however, the large-domain results indicate improved resolution of the flow close to the hull and wake centerplane and of the Froude number differences due to near-wall turbulence modeling and nonlinear free-surface boundary conditions. Additional evaluation is provided through discussion of the recent CFD Workshop Tokyo 1994, where both methods were among the best. Last, some concluding remarks are made. © 1996 Academic Press, Inc.

INTRODUCTION

There is a current need for the development and validation of computational fluid dynamics (CFD) methods for surface-ship boundary layers and wakes and wave fields in support of anticipated future designs. At present, both interactive and large-domain approaches for solving the Reynolds-averaged Navier-Stokes (RANS) equations are viable alternatives and need relative assessment. The former refers to interactively combining viscous- and inviscidflow methods for the inner and outer regions, respectively, whereas the latter refers to using a viscous-flow method for the entire domain. The interactive approach has the advantages of both the inviscid- and viscous-flow methodologies and computational efficiency, but it is inherently limited by the inviscid-flow method. The large-domain approach has the advantages of ease of implementation, more general applicability (e.g., massively separated flows), and utility (e.g., inclusion of thermal stratification, two-fluid modeling, etc.) and the limitations solely due to Reynolds

averaging and current pacesetting issues in the CFD development. Also, both approaches are subject to developments associated with the treatment of free-surface boundary conditions and effects. Here again, the largedomain approach has an advantage in the ability for inclusion of viscous and turbulence effects over the entire domain. Thus, the large-domain approach is ultimately superior in offering more general applicability and utility and the possibility of a more detailed resolution of the flow.

The authors have been involved in the development of both approaches. Initially, an interactive approach was developed and validated [1, 2]. Herein, modifications and extensions are made for a large-domain approach, including improved numerical algorithms, Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. The results are validated through comparisons with data for the Series $60 C_{\rm B} = 0.6$ ship model at low and high Froude numbers (Fr = U_0/\sqrt{gL}) [3, 4], and the precursory interactive approach. The latter comparisons also enable a qualitative relative assessment of the two approaches. Cost and time constraints, as well as problem complexity, precluded a quantitative relative assessment as was possible previously for zero Fr [5]. The zero Fr solution refers to the treatment of the free surface as a symmetry plane (i.e., double-body solution) corresponding to a condition of infinite gravity. Additional evaluation is provided through discussion of the performance of both approaches at the recent CFD Workshop, Tokyo 1994 [6], where both methods were among the best, along with a summary and conclusions from the workshop, which also serve as an update of the reviews of [1, 2].

In the following, a detailed overview of the computational method is provided. The conditions and grids are described. Then, the results are validated for Fr = 0and 0.316 through comparisons with the data and the interactive approach. Next, the *CFD Workshop*, *Tokyo* 1994 [6] is discussed. Last, some concluding remarks are made.

^{*} Current address: Department of Marine System Engineering, College of Engineering, University of Osaka Prefecture, 1-1, Gakuen-cho, Sakai, Osaka 593, Japan.

COMPUTATIONAL METHOD

As already noted, the computational method is based on modifications and extensions of the precursory interactive approach [1, 2]. Recently, Stern *et al.* [9] provided a detailed description of the method to aid in the transition for design applications, including extensions for naval combatants with bulbous bows and transom sterns, utilizing multiblock domain decomposition and propeller-hull interaction, utilizing the method of Stern *et al.* [10]. Herein, an overview is given. Note that the core viscous-flow solver is based on Chen and Patel [7, 8] which has been validated for a variety of benchmark cases and applications.

Equations, Coordinate Systems, and the Turbulence Model

The unsteady RANS and continuity equations for an incompressible fluid are written in nondimensional form and Cartesian tensor notation as

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial \hat{p}}{\partial x_i} + \frac{1}{\operatorname{Re}} \frac{\partial^2 U_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} (\overline{u_i u_j}), \quad (2)$$

where $U_i = (U, V, W)$ are the mean-velocity components, $x_i = (X, Y, Z)$ are the Cartesian coordinates, \hat{p} is the piezometric pressure $(p + \rho g z)$, $\overline{u_i u_j}$ are the Reynolds stresses, ν is the kinematic viscosity, and Re = $U_o L/\nu$ is the Reynolds number. The Reynolds stresses are related to the mean rate of strain through an isotropic eddy viscosity ν_t ,

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \,\delta_{ij} k,\tag{3}$$

where δ_{ij} is the Kronecker delta and k is the turbulent kinetic energy. The equations are normalized by reference velocity U_o and length L and density ρ . Closure is attained through the Baldwin–Lomax algebraic turbulence model without modifications for free-surface effects.

The equations are transformed into nonorthogonal curvilinear coordinates such that the computational domain forms a simple rectangular parallelepiped with equal grid spacing. The transformation is a partial one since it involves the coordinates, only, and not the velocity components U_i . The transformation is accomplished through use of the expression for the divergence and "chain-rule" definitions of the gradient and Laplacian operators which relate the Cartesian coordinates x_i to the nonorthogonal coordinates (ξ, η, ζ) . In this manner, (1) and (2) with (3) can be written in the form

$$\frac{\partial U^i}{\partial \xi^i} = 0 \tag{4}$$

$$g^{ii}\frac{\partial^2 \phi}{\partial \xi^i \partial \xi^i} - 2a^j_{\phi}\frac{\partial \phi}{\partial \xi^j} = R_{\rm eff}\frac{\partial \phi}{\partial \tau} + S_{\phi},\tag{5}$$

where U^i are the modified contravariant velocity components in the direction of the transformed coordinates, $\phi = (U, V, W)$, R_{eff} is an effective Reynolds number $(1/R_{\text{eff}} = 1/\text{Re} + \nu_i)$, S_{ϕ} are source terms, τ is the computational time, and the geometric coefficients b_i^j , g^{ii} , and Jacobian J are functions of the coordinates only.

Discretization and Pressure Equation

Equation (5) is reduced to algebraic form through the use of the finite-analytic method. Solution-dependent coefficients are analytically derived by solving the linearized momentum equation on a computational cell with dimensions $21 \times 2k \times 2h$ using a hybrid method which combines a two-dimensional analytic solution in the $\eta\zeta$ -plane with a one-dimensional analytic solution in the ξ -direction. By specifying boundary conditions on the faces of the cell as a combination of exponential and linear functions, which are the natural solutions for the linearized momentum equation, (5) can be solved by the method of separation of variables. When the solution is evaluated at the center node P (i.e., (i, j, k)) of the element, the following 12-point finite analytic formula is obtained,

$$\phi_{P}^{n} = \frac{1}{1 + C_{P}(C_{U} + C_{D} + R_{\text{eff}}/\Delta\tau)} [C_{NE}\phi_{NE}^{n} + C_{NW}\phi_{NW}^{n} + C_{SE}\phi_{SE}^{n} + C_{SW}\phi_{SW}^{n} + C_{EC}\phi_{EC}^{n} + C_{WC}\phi_{WC}^{n} + C_{NC}\phi_{NC}^{n} + C_{SC}\phi_{SC}^{n} + C_{P}(C_{U}\phi_{U} + C_{D}\phi_{D} + R_{\text{eff}}\phi_{P}^{n-1}/\Delta\tau - (S_{\phi})_{P})],$$
(6)

where C_{nb} (nb = NE, NW, etc.), C_P , C_U , and C_D are the finite-analytic coefficients and the subscripts P, U, and Ddenote the center, upstream, and downstream nodes, respectively, and NC, NW, WC, etc. denote the nodes in the $\eta\zeta$ -plane in terms of compass directions. The superscripts (n) and (n - 1) refer to the current and previous time levels and $\Delta\tau$ is the time step. It is seen that ϕ_p depends on all eight neighboring nodal values in the transverse plane, as well as the values at the upstream and downstream nodes ϕ_U and ϕ_D and the values at the previous time step ϕ_P^{n-1} . The pressure-gradient and cross-derivative terms in $(S_{\phi})_P$ are evaluated using second-order central finite differences. A regular-grid (i.e., variable-collocated) approach is used such that (6) is evaluated at P for $\phi =$ U, V, and W.

The use of a regular grid results in the problem of odd– even decoupling of the pressure and velocity fields. Therefore, the derivation of the pressure equation requires special treatment to eliminate this problem. The pressure equation is derived using the generalized continuity equation (4) in discrete form and a staggered-grid control volume. Discretizing (4) on this control volume using secondorder central finite differences, decomposing the modified contravariant velocity into pseudovelocity and pressuregradient terms, and discretizing the pressure-gradient terms with second-order central finite differences about the staggered-grid locations results in

$$(E_d^{11} + E_u^{11} + E_n^{22} + E_s^{22} + E_e^{33} + E_w^{33})\hat{p}_P = E_d^{11} \hat{p}_D + E_u^{11} \hat{p}_U + E_n^{22} \hat{p}_{NC} + E_s^{22} \hat{p}_{SC}$$
(7)
 $+ E_e^{33} \hat{p}_{EC} + E_w^{33} \hat{p}_{WC} - \hat{D},$

where E^{ij} and \hat{D} are the pressure coefficients and mass source, respectively, and are defined as

$$E^{ij} = \frac{R_{\rm eff}C_P}{J[1 + C_P(C_U + C_D + R_{\rm eff}/\Delta\tau)]} b^i_m b^j_m \qquad (8)$$

$$\hat{D} = \hat{U}_d^1 - \hat{U}_u^1 + \hat{U}_n^2 - \hat{U}_s^2 + \hat{U}_e^3 - \hat{U}_w^3, \tag{9}$$

where \hat{U}^i is a modified contravariant pseudovelocity field which contains part of the pressure-gradient if the grid is nonorthogonal. Since (7) is in terms of E^{ij} and \hat{D} at the staggered-grid locations, one-dimensional linear interpolation is used to obtain both of these quantities at the staggered nodes, but in terms of the values calculated at the regular-grid locations.

Boundary Conditions and Grid Generation

Referring to Fig. 1, the specified boundaries of the solution domain are the body surface S_b , the inlet plane S_i , the exit plane S_e , the symmetry plane S_k , the outer boundary S_o , and the free-surface S_{ζ} . Note that for zero Fr, S_{ζ} becomes the waterplane S_w .

For zero Fr the boundary conditions are as follows: on S_b , $\phi = \partial p/\partial n = 0$ (where *n* is normal to the body); on S_i , free-stream values are imposed, i.e., $U = U_0$, V = W = p = 0; on S_e , a zero gradient condition is used, i.e., $\partial(\phi, p)/\partial X = 0$; on S_k and S_w , $\partial(U, W, p)/\partial Y = V = 0$ and $\partial(U, V, p)/\partial Z = W = 0$, respectively; and on S_o , $U = U_0$, $V = \partial p/\partial r = 0$ (where *r* is the radial coordinate), and *W* is extrapolated.

For nonzero Fr, the boundary conditions are similar, except: p is replaced by \hat{p} ; and on S_{ξ} , exact nonlinear kinematic and approximate dynamic free-surface conditions are applied on the actual free surface, which is determined as part of the solution. The derivation of these conditions and the procedure for determining the free surface are described next. On the free surface, there are two boundary conditions. The kinematic condition requires that ζ is a stream surface:

$$\frac{DF}{Dt} = 0, \quad F = \zeta - z. \tag{10}$$

The dynamic condition requires that the normal and tangential stresses are continuous across the surface,

$$\tau_{ij}n_j = \tau^*_{ij}n_j,\tag{11}$$

where n_j is the unit normal vector to the free surface and $\tau_{ij} (=-p\delta_{ij} + \text{Re}^{-1}(\partial U_i/\partial x_j + \partial U_j/\partial x_i) - \overline{u_i u_j})$ and τ_{ij}^* are the fluid- and external-stress tensors, respectively. The latter includes surface tension. The following approximations were made in employing (11): (a) the external stress and surface tension are assumed to be zero; (b) the curvature of the free surface and gradients of the normal velocity component are assumed small. Expansion of (10) and reduction of (11) results in

$$\frac{\partial \zeta}{\partial t} + \tilde{U}\frac{\partial \zeta}{\partial \xi} + \tilde{V}\frac{\partial \zeta}{\partial \eta} - W = 0, \qquad (12)$$

where

$$\tilde{U} = \frac{1}{J} (b_1^1 U + b_2^1 V)$$
$$\tilde{V} = \frac{1}{J} (b_1^2 U + b_2^2 V)$$
$$\hat{p} = \frac{\zeta}{\mathrm{Fr}^2}$$
(13)

$$\frac{\partial(U,V)}{\partial z} = 0. \tag{14}$$

Last, a zero-gradient condition is used for W, which is consistent with the approximations employed for the dynamic condition,

$$\frac{\partial W}{\partial z} = 0. \tag{15}$$

Note that the pressure boundary condition (13) is inviscid in that it can be obtained directly by neglecting the normal viscous stress.

These conditions were implemented as follows: (13)–(15) are used as boundary conditions for the pressure and velocity; Eq. (12) is used to solve for the free-surface ζ iteratively, in conjunction with the global iteration procedure. For each global iteration, (12) is solved, whereupon the computational grid is regenerated so that it conforms



FIG. 1. Solution domain and computational grid: (a) Longitudinal plane; (b) Body crossplane.

to both the body and the free-surface. The procedure for solving (12) and adjusting the grid is outlined next.

Equation (12) is solved using a Beam and Warming linear-multistep method based upon space-centered finitedifferences, approximate factorization, and the addition of both implicit and explicit fourth-order artificial dissipation. The temporal derivative is expressed as

$$\zeta^{n+1} = \zeta^n + \frac{\Delta t}{2} [(\zeta_t)^n + (\zeta_t)^{n+1}]$$
(16)

or

$$\zeta^{n+1} = \zeta^n - \frac{\Delta t}{2} [(\tilde{U}\zeta_{\xi} + \tilde{V}\zeta_{\eta} - W)^n + (\tilde{U}\zeta_{\xi} + \tilde{V}\zeta_{\eta} - W)^{n+1}].$$
(17)

Introducing the delta form such that $\Delta \zeta^n = \zeta^{n+1} - \zeta^n$ and the differential operator L, (17) reduces to

$$L[\Delta \zeta^n] = \Delta t (W - \tilde{U}\zeta_{\xi} - \tilde{V}\zeta_{\eta})^n, \qquad (18)$$

where

$$L = \left\{ 1 + \frac{\Delta t}{2} \left(\frac{\partial}{\partial \xi} (\tilde{U})^n + \frac{\partial}{\partial \eta} (\tilde{V})^n \right) \right\}.$$
 (19)

If L is factored into one-dimensional operators, L_1 and L_2 , (18) becomes

$$L_1[\Delta \zeta^n] \cdot L_2[\Delta \zeta^n] = \Delta t (W - \tilde{U}\zeta_{\xi} - \tilde{V}\zeta_{\eta})^n, \qquad (20)$$

where

-181 -

$$L_{1} = \left\{ 1 + \frac{\Delta t}{2} \frac{\partial}{\partial \xi} (\tilde{U})^{n} \right\}$$

$$L_{2} = \left\{ 1 + \frac{\Delta t}{2} \frac{\partial}{\partial \eta} (\tilde{V})^{n} \right\}.$$
(21)

Equation (20) can now be solved in two one-dimensional inversions

Step 1.
$$L_1[\Delta \zeta'] = \Delta t (W - \tilde{U}\zeta_{\xi} - \tilde{V}\zeta_{\eta})^n$$
 (22)

$$Step 2. L_2[\Delta \zeta^n] = \Delta \zeta'.$$
⁽²³⁾

Finally, the spatial derivatives in L_1 and L_2 are discretized using a second-order central difference and both implicit and explicit fourth-order artificial dissipation ε are added to damp out oscillations and/or maintain stability:

Step 1.
$$L_{1}^{*}[\Delta \zeta'] = \Delta t \left(W - \tilde{U}\zeta_{\xi} - \tilde{V}\zeta_{\eta} - \varepsilon_{\xi} \frac{\partial^{4}\zeta}{\partial \xi^{4}} - \varepsilon_{\eta} \frac{\partial^{4}\zeta}{\partial \eta^{4}} \right)^{n}$$
 (24)

$$Step \ 2. \ L_2^*[\Delta \zeta^n] = \Delta \zeta', \tag{25}$$

where

$$L_{1}^{*} = \left\{ 1 + \frac{\Delta t}{2} \frac{\partial}{\partial \xi} (\tilde{U})^{n} + \varepsilon_{\xi} \frac{\partial^{4}}{\partial \xi^{4}} \right\}$$

$$L_{2}^{*} = \left\{ 1 + \frac{\Delta t}{2} \frac{\partial}{\partial \eta} (\tilde{V})^{n} + \varepsilon_{\eta} \frac{\partial^{4}}{\partial \eta^{4}} \right\}.$$
(26)

The amount of artificial dissipation is $\varepsilon = 0.4$ and local time stepping is used with $\Delta t = 0.001(U^2 + V^2 + W^2)^{-1/2}$. Parametric studies indicate convergent solutions for $0.04 \le \varepsilon \le 4$. Smaller values produce checkerboard decoupling of the wave solution, whereas large values cause excessive wave damping. For each step in the approximate factorization (24) and (25), a pentadiagonal solver is used to determine the wave solution. Referring to Fig. 1, (12) is solved with boundary conditions: on the inlet boundary, $\Delta \zeta = 0$; on the exit boundary, $\Delta \zeta = ax^2 + c$; on the body and centerline boundaries, $\Delta \zeta = aY^2 + c$; and on the farfield boundary, $\partial (\Delta \zeta) / \partial Y = 0$. Finally, to accurately resolve both the wave and velocity fields, a method was developed to solve (12) using a two-dimensional grid generated separately from, but coupled to, the RANS grid. A simple procedure is used to facilitate the conforming of the RANS grid to both the hull and the free surface.

The RANS grid is H-type with constant-X planes

stacked to form a complete three-dimensional grid. The bow and stern are resolved with axial clustering of grid points which are distributed using hyperbolic tangent stretching functions. Because of the H-type grid, non-vertical bows and sterns are resolved in a staircase fashion. The constant-X cross-plane grids are generated elliptically by solving a Poisson equation for the transformation between (Y, Z) and (η, ζ) . Spacings are specified in the η -direction at the surface of the hull, which for the Baldwin-Lomax turbulence model should be at a $Y^+ \approx 1$, and in the ζ direction at both the centerplane and free surface. The initial grid must extend to an elevation sufficiently above the zero, or design waterline, to allow for wave crests. As the wave field develops, the RANS grid conforms to the free surface. By confining the grid points and saving the initial distribution along $\eta = \text{const}$, or girth-wise lines, the grid is easily updated; the point on the free surface moves to a new elevation and all points below the free-surface slide along the η = const line so as to maintain the initial relative distribution.

The kinematic free-surface boundary condition (12) grid is two-dimensional (i.e., function of (X, Y)), updated iteratively to fit the wave-hull intersection, and it is different from the RANS grid in that, instead of high near-wall resolution, more points are distributed in the outer flow to resolve the wave field. The grid is 441 × 105 and consists of equal spacing in the axial direction and a power distribution

$$Y_{i,j} = (Y_{\text{body}} + 0.0002) + (1 - Y_{\text{body}}) \left(\frac{j-1}{jmax - 1}\right)^n \quad (27)$$

in the transverse direction, where n = 2, jmax is number of points, and 0.0002 is an offset to ensure nonzero velocity. For regions upstream and downstream of the ship, $Y_{\text{body}} = 0$.

Communication between the RANS and free-surface grids is accomplished using bi-linear interpolation such that the velocity field from the $\zeta = \text{kp1}$ plane of the RANS grid is interpolated to the free-surface grid. Similarly, the wave elevation is interpolated from the free-surface grid to the $\zeta = \text{kp1}$ plane of the RANS grid.

Overall Solution Procedure

The overall solution algorithm is based on the pressureimplicit split-operator (PISO) algorithm, where the velocity and pressure fields are coupled through a two-step iterative procedure. In the first step, the momentum equations (6) are solved implicitly, using a tridiagonal algorithm and the method of lines and the pressure from the previous time step (n-1) for an intermediate velocity field denoted by an asterisk:

$$\phi_P^* = \frac{1}{1 + C_P (C_U + C_D + R_{\text{eff}} / \Delta \tau)} [C_{NE} \phi_{NE}^* + C_{NW} \phi_{NW}^* + C_{SE} \phi_{SE}^* + C_{SW} \phi_{SW}^* + C_{EC} \phi_{EC}^* + C_{WC} \phi_{WC}^* + C_{NC} \phi_{NC}^* + C_{SC} \phi_{SC}^* + C_P (C_U \phi_U^* + C_D \phi_D^* + R_{\text{eff}} \phi_P^{n-1} / \Delta \tau - (S_{\phi})_P)].$$
(28)

For steady flow, convergence of (28) is not required; therefore, only several iterations are used.

The second step consists of subiterations. First, the pressure equation (7) is solved implicitly, using a tridiagonal algorithm and the method of lines and ϕ_p^* for an intermediate pressure denoted by an asterisk:

$$(E_d^{11} + E_u^{11} + E_n^{22} + E_s^{22} + E_e^{33} + E_w^{33})\hat{p}_P^*$$

= $E_d^{11}\hat{p}_D^* + E_u^{11}\hat{p}_U^* + E_n^{22}\hat{p}_{NC}^* + E_s^{22}\hat{p}_{SC}^*$ (29)
+ $E_e^{33}\hat{p}_{EC}^* + E_w^{33}\hat{p}_{WC}^* - \hat{D}^*.$

For steady flow, convergence of (29) is not required; therefore, only several iterations are used. Second, the momentum equation (29) is solved explicitly using \hat{p}^* for the momentum correction denoted by a double asterisk:

$$\phi_P^{**} = \frac{1}{1 + C_P (C_U + C_D + R_{\text{eff}} / \Delta \tau)} \left[C_{NE} \phi_{NE}^{*} + C_{NW} \phi_{NW}^{*} + C_{SE} \phi_{SE}^{*} + C_{SW} \phi_{SW}^{*} + C_{EC} \phi_{EC}^{*} + C_{WC} \phi_{WC}^{*} + C_{NC} \phi_{NC}^{*} + C_{SC} \phi_{SC}^{*} + C_P (C_U \phi_U^{*} + C_D \phi_D^{*} + R_{\text{eff}} \phi_P^{n-1} / \Delta \tau - (S_\phi)_P) \right].$$
(30)

 ϕ_P^{**} is then used to update \hat{p}_P^* . For steady flow, convergence of the second step is not required; therefore, only several subiterations between (29) and (30) are used. At the completion of the second step, \hat{p}_P^* and ϕ_P^{**} are underrelaxed and become \hat{p}_P^n and ϕ_P^n . Convergence is determined using the L_2 norms (residuals) of the solution variables between time steps (n) and (n-1),

$$\operatorname{RES}_{\phi} = \frac{\sum_{i=1}^{ip1} \sum_{j=1}^{kp1} \sum_{k=1}^{kp1} |\phi_{i,j,k}^n - \phi_{i,j,k}^{n-1}|}{ip1 * jp1 * kp1},$$
(31)

where ϕ represents U, V, W, \hat{p} , or ζ and should ideally display three or four orders of magnitude drop for a converged solution.

CONDITIONS AND GRIDS

The conditions simulate the experiments; i.e., for low Fr, Fr = 0 and Re = 2×10^6 , and for high Fr, Fr =

0.316 and Re = 4×10^6 . Note that in the former case the experimental Fr = 0.16; however, free-surface effects are negligible, except near the bow. A partial view of the Fr = 0.316 RANS grid is shown in Fig. 1. For both Fr, the inlet, exit, and outer boundaries are located at X = (-0.4,2.0) and r = 1. The first grid points off the body surface are located in the range $Y^+ < 2$ (=Re $U_\tau Y_n$). The RANS grid sizes were: for Fr = 0, $90 \times 30 \times 20 = 54000$; and for Fr = 0.316, $180 \times 40 \times 30 = 216000$ and $360 \times 60 \times 10^{-10}$ 30 = 648000. In the latter case, although the grid number was increased by a factor of 3, the differences between its solution and the former were relatively small (e.g., 4% change in C_T). The free-surface boundary condition grid size was 460×100 . The values of the time increment and underrelaxation factors for velocity and pressure are 0.01, 1, and 0.1. The code has been optimized for vector machines and has demonstrated performance of 150+ Mflops (millions of floating-point operations per second) on the CRAY C90. C90 CPU-time and memory requirements were about 0.35 h and 4 MW (megawords) for Fr = 0 and 16.7/5.6 h and 43/14.3 MW for the two Fr = 0.316 grids. In the latter cases, this corresponds to approximately 4.6×10^{-5} s/grid-point/global-iteration and increases by factors of about 14/5 and 29/10 in comparison with the interactive approach. The convergence criterion was that the residual (31) for all variables be about 10^{-4} , which was satisfied in about 500 and 2000 global iterations for Fr = 0 and 0.316. Additional global iterations indicated small oscillations (i.e., 2% C_T) with minimal reduction in the residuals.

Note that recent work with the method includes extensive verification analysis (iterative and grid convergence and order-of-accuracy studies) for both simple and practical geometries [9]. The status is as follows: the uncertainty in iterative convergence is about 2%, i.e., the same as the present results; the uncertainty in grid convergence is about 1-2%, i.e., better than the present results due to the use of somewhat finer grids; and order of accuracy's vary between about 1.5-2.5 depending on geometry and flow complexity.

RESULTS

Results are presented in Figs. 2–7 for both Fr = 0 and 0.316; however, the emphasis is on the latter. The discussions focus on comparisons with both the data and interactive approach based on all data locations (see Fig. 3 of [3]), although only representative results are shown. For brevity, in most cases, the data and interactive approach results are not reproduced, but rather, a similar format is used as in [2–4].

For Fr = 0 (Figs. 6–7), the general features of the Fr = 0.16 data are predicted; i.e., stagnation effects near the bow, thin boundary-layer development on the forebody, and rapid thickening of the boundary layer on the



FIG. 2. Wave profiles.

afterbody with a characteristic bulge in the axial-velocity contours in the midgirth region and weak longitudinal vortex. Close to the hull and wake centerplane, improvement over [2] is indicated, due to the superior performance of the present Baldwin–Lomax near-wall turbulence model versus the previous k- ε turbulence model with wall functions. Apparently, the effects of near-wall modeling override zero versus two-equation differences. However, despite this improvement, the assessment is the same as [2]; i.e., significant improvements are required in the CFD development for detailed resolution of the viscous flow.

For Fr = 0.316, the wave profile (Fig. 2) shows overall close agreement with the data, including amplitude, shape, and phase; however, at the bow, the amplitude is underpredicted. Subsequent work has shown that this discrepancy



FIG. 3. Wave-elevation and wave-slope contours.

is due to the inability of the CFD to simulate a thin film (about 1.5-mm thick) and bow vortex present at the bow of the Series 60 $C_B = 0.6$ ship model [11]. The waveelevation and -slope contours (Fig. 3) also show similarity with the data, especially in the local region; however, differences are observed in the global region. The differences are highlighted by the transverse and longitudinal waveelevation profiles (Fig. 4). In the former case (Fig. 4a), on the forebody, good agreement is indicated with both the local and global data, although the bow-wave crest and shoulder-wave trough are somewhat under and overpredicted. On the afterbody, good agreement is also indicated with the local data; however, as already noted, the complex global-region wave system is not replicated. In the latter case (Fig. 4b), the results are consistent, i.e., the amplitudes of the crests and troughs are underpredicted, especially in the global region. The discrepancies increase with distance from the hull. Improvement over [2] is indicated, especially for the wave profile and local-region wave system, due to the inclusion of nonlinear effects. Both approaches display a similar inability to predict the global-region wave system, which is unexpected due to the large methodological differences between the two approaches in this region.

The surface pressure and gradient contours (Fig. 5) show good agreement with the data with regard to regions of positive and negative pressure and favorable and adverse pressure gradient. The results are similar to [2], except for nonlinear effects. The wall-shear stress contours (figure not shown) are consistent with expectation; however, no data is available for comparison. The value for the total resistance coefficient C_T was 5.79 \times 10⁻³, which is within 2% of the data ($C_T = 5.89 \times 10^{-3}$). Note that this percent difference is at about the accuracy of the data 0.5-5%. The value for the calculated pressure-resistance coefficient C_P was 2×10^{-3} , whereas the value for the experimental residuary-resistance coefficient C_R (i.e., C_T – $C_{F0})$ was 2.462 \times 10^{-3} (i.e., -19% difference). The value for the frictionalresistance coefficient C_F was 3.8 \times 10⁻³, whereas the Schoenherr coefficient C_{F0} is 3.423×10^{-3} (i.e., 11% difference, which is equivalent to a form factor k = 0.11).

The mean-velocity and pressure field results (Figs. 6–7) show:

at X = 0.0, the general features of the stagnation effects displayed in the data. The U and \hat{p} contours show close similarity to the data with respect to regions of low velocity and high pressure. The U contours are curved back toward the centerplane as is observed in the data. The velocity and pressure profiles show similar Fr differences and good agreement with the data, except that the V values are somewhat underpredicted, which may be due to lack of grid resolution with regard to finite-thickness effects. The U values are larger for Fr = 0.316, but the differences reduce with depth. The V profiles show small Fr differ-



FIG. 4. Wave-elevation profiles: (a) Transverse; (b) Longitudinal.

-185-



FIG. 5. Surface-pressure and gradient contours: Pressure (upper); Axial gradient (middle); Vertical gradient (lower).

ences, whereas the W values are larger for Fr = 0.316 for all depths. The \hat{p} profiles are similar to the U profiles, but with a reverse trend in magnitude.

At X = (0.1, 0.2, 0.4), the free-surface effects on the thin boundary-layer development are simulated. The U contours are similar to the data with regard to the broad region of U < 1 at X = 0.1 and 0.2 and increased values in the outer region at X = 0.4. This is also true for the \hat{p} contours regarding the broad region of high pressure at X = 0.1 and 0.2. At X = 0.4, the \hat{p} contours show lower values for Fr = 0.316 and the contours are nearly parallel to the free surface as is observed in the data. The velocity and pressure profiles show similar trends to the data, including reduced Fr differences with depth. At X = 0.1 and 0.2, the U profiles indicate smaller values for Fr = 0.316, whereas these show larger values at X = 0.4. At X = 0.1and 0.2, the V values are reduced and increased in the inner and outer regions. At X = 0.4, the V profiles show minimal Fr differences. At X = 0.1, the W profiles show reduced and increased values in the inner and outer regions, whereas they show reduced values overall at X =0.2 and 0.4. These complex features of the flow are due to the formation of the bow-bilge vortex.

At X = (0.6, 0.8, 0.9), free-surface effects on the thick boundary-layer development are simulated. At X = 0.6, the U contours show reduced values in the outer region; and, at X = 0.8 and 0.9, the reduction in the boundary layer and more pronounced mid-girth bulge is observed for Fr = 0.316, which are also shown in the data. The \hat{p} contours show similar trends to the data; i.e., the regions of increased and decreased values are reproduced. The formation of the stern-bilge vortex is reproduced, however, with smaller magnitude than the data. The velocity and pressure profiles show similar trends to the data. The Fr differences for these profiles penetrate to larger depths as is observed in the data. However, the V and W profiles generally have smaller predicted values in the inner region.

At X = (1, 1.1, 1.2), the free-surface effects on the wake development are simulated. The U and \hat{p} contours show similarity to the data; however, they lack detail, especially near the core of the longitudinal vortex. The depthwise reduction in the boundary-layer thickness for Fr = 0.316is observed in the U contours, as is the case for the data. The vectors show that the longitudinal vortex is shifted outward as is seen in the data; however, the magnitudes are underpredicted. The velocity and pressure profiles show similar trends to the data; however, some discrepancies are observed in the U profiles near the centerplane at larger depths. This is also true for the V and W profiles. These shortcomings are similar to those for Fr = 0. On the other hand, the \hat{p} values show good agreement with the data. The results show improvement over [2] with regard to the resolution of the flow close to the hull and wake centerplane and of the Fr differences, due to near-wall turbulence modeling and nonlinear effects; however, detailed resolution of the viscous flow is still lacking.

CFD WORKSHOP TOKYO 1994

Recently, the CFD Workshop, Tokyo 1994 [6] was held in Tokyo, Japan on March 22-24, 1994. The purpose of the workshop was to assess the state of the art of CFD for steady flow around ships and to accelerate further developments through discussion among the participants, in conjunction with comparisons of the computed results with each other and with the data. Previous related workshops were held on viscous flow in 1980 and 1990 [12, 13] and on inviscid flow in 1978, 1982, and 1987 [14-16]. Three test cases were used: (1) turbulent free-surface flow (Series 60 $C_B = 0.6$ ship model); (2) inviscid free-surface flow (Series $60 C_B = 0.6$ ship model); and (3) turbulent double-model flow (HSVA ship models). Both the present large-domain and precursory interactive results were included in Test Case 1. Note that the data used presently [3, 4] was also used at the workshop for Test Cases 1 and 2.

In Test Case 3, 15 entries representing 7 nations participated. The CFD methods included both pseudo-compress-



FIG. 6. Axial-velocity and pressure contours and crossplane vectors: (a) x = 0; (b) x = 0.2, (c) x = 0.6; (d) x = 0.9; (e) x = 1.1.

ibility and SIMPLER, PISO, and MAC types with firstand higher-order discretization procedures, simple and advanced grid-generation techniques, and isotropic (Cebici-Smith, Baldwin-Lomax, and k- ε) and nonisotropic (differential Reynolds-stress) turbulence models. The Baldwin-Lomax and k- ε models included the use of both wall functions and near-wall models and, in some cases, modifications for the present applications. Also, in one case, a very simplified subgrid scale model was used. The comparisons indicated that the nonmodified isotropic models were unable to produce the detailed flow pattern (e.g., hook-shaped axial-velocity contours near the stern), similarly as in the previous related workshop. The modified Baldwin-Lomax and k- ε models with near-wall models showed some improvement, but were insufficiently documented, including physical justification, to be considered generally useful. However, the differential Reynolds-stress models showed marked improvement, which confirms the necessity for nonisotropic turbulence models for the present applications. Also, higher-order discretization procedures and advanced grid-generation techniques indicated improvement.

In Test Case 2, eight entries representing seven nations

participated. The CFD methods included potential and Euler types utilizing both linear and nonlinear free-surface boundary conditions. In many aspects, the inviscid-flow results showed remarkably good agreement with the data, including wave profile, surface-pressure distribution, and residuary resistance. However, the wave profiles indicated a downstream phase shift, the stern pressure rise and waves were overpredicted, the detailed wave pattern associated with the complex interactions of the bow, shoulder, and stern waves was not accurately resolved, especially the wave slopes, and the viscous effects were completely missing. The potential and Euler types showed similar results with the nonlinear methods indicating improvement; however, definitive general conclusions cannot yet be reached as to the differences between the potential versus Euler and linear versus nonlinear methods.

In Test Case 1, 11 entries representing 4 nations participated. The CFD methods were based on extensions of the types described for Test Case 3 for nonzero Fr calculations utilizing both linear and nonlinear approximate free-surface boundary conditions and interactive and large-domain approaches. In comparison to Test Case 3, relatively fewer methods were able to obtain satisfactory results. Both apTAHARA AND STERN



rig. 0—Commueu

CALCULATING SHIP BOUNDARY LAYERS



FIG. 7. Velocity and pressure profiles: (a) x = 0; (b) x = 0.2; (c) x = 0.6; (d) x = 0.9; (e) x = 1.1.

proaches showed similar results with regard to free-surface effects as described for Test Case 2, but with significant improvement for the wave profile phase, stern pressure rise and waves, and the presence of viscous effects. The resistance values were at about the accuracy of the data. Also, viscous effects for the Series 60 $C_B = 0.6$ ship model for nonzero Fr are similar to those for zero Fr, but with wave-induced pressure gradient effects, such that the conclusions with regard to Test Case 3 are also valid for Test Case 1.

In all three test cases, the entries showed varying capabilities and, in many cases, their procedures were insufficiently documented. In particular, limited verification analysis (i.e., iterative and grid convergence and order of accuracy studies) was provided. The previous discussion was based on the best quality entries, which, in Test Case 1, included both the present large-domain and precursory interactive approaches. The workshop was successful with the conclusion that the status of CFD for ship hydrodynamics is such that the predicted steady resistance and flow are nearly at the accuracy of the experimental data. However, certain detailed flow features of the wave pattern and viscous flow are inadequately resolved, indicating the need for improvements in numerical accuracy, turbulence modeling (e.g., use of nonisotropic turbulence models), grid generation, and treatment of the free-surface boundary conditions. Nonetheless, the status suggests that the better methods will be useful for design applications. The results from such applications should also be useful in aiding improvements, which are expected to follow in conjunction with extensions for more complex flows.

CONCLUDING REMARKS

A large-domain approach is developed for calculating ship boundary layers and wakes and wave fields for nonzero Fr. The RANS and continuity equations are solved with the Baldwin–Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. The results are validated through comparisons with data for the Series 60 $C_B = 0.6$ ship model at low and high

409



FIG. 7—Continued

Fr and results of the precursory interactive approach. Both approaches yield satisfactory results; however, the large-domain results indicate improved resolution of the flow close to the hull and wake centerplane and of the Fr differences due to near-wall turbulence modeling and nonlinear free-surface boundary conditions. Both methods were among the best at the recent *CFD Workshop, Tokyo 1994* [6].

Although both approaches accurately predict the general features of the flow, detailed resolution is lacking both for the wave field and viscous flow. For the interactive approach, improvements require inclusion of nonlinear effects and in CFD development. Based on [17], the former is not expected to significantly improve the detailed predictions. For the large-domain approach, improvements require development of procedures for more accurate imposition of the free-surface boundary conditions, including inclusion of viscous and turbulence effects and in CFD development. Based on [18], the viscous terms are expected to have a significant influence close to the free surface in regions of large velocity and wave gradients. Improvements in CFD development require advancements in higher-order discretization procedures, turbulence modeling, and grid generation.

As already noted, improvements in CFD development are expected to follow in conjunction with extensions for more complex flows. For example, recent work with the present method includes extensions for unsteady flow with application to wave-induced separation for a surface-piercing NACA 0024 foil over a range of Fr [19] and for the inclusion of thermal-stratification and two-fluid modeling with application to near- and far-field flows for naval combatants [20]. Hopefully, through such extensions and applications, in conjunction with verification, validation, and calibration methods such as the present one, will be accredited as engineering tools for future designs.

ACKNOWLEDGMENTS

This research was sponsored by the Office of Naval Research Grant N00014-92-1092 under the administration of Dr. E. P. Rood. The computations were performed on the Army Waterways Experiment Station and National Aerodynamic Simulation Program supercomputers. Dr. E. G. Paterson is gratefully acknowledged for his help in preparing the manuscript.

REFERENCES

- 1. Y. Tahara, F. Stern, and B. Rosen, J. Comput. Phys. 98(1), 33 (1992).
- 2. Y. Tahara and F. Stern, J. Comput. Fluids 23(6), 785 (1994).
- 3. Y. Toda, F. Stern, and J. Longo, J. Ship Res. 37(4), 360 (1992).
- 4. J. Longo, F. Stern, and Y. Toda, J. Ship Res. 37(1), 16 (1993).
- 5. F. Stern, S. Y. Yoo, and V. C. Patel, AIAA J. 26(9), 1052 (1988).
- 6. Proceedings, CFD Workshop Tokyo, Ship Research Institute Ministry of Transport Ship & Ocean Foundation, 22–24 March 1994, Vols. 1, 2.
- H. C. Chen and V. C. Patel, "The Flow around Wing-Body Junctions," in 4th Sympos. Numer. Phys. Aspects Aerodyn. Flows, Long Beach, CA, 1989 (unpublished).
- 8. H. C. Chen and V. C. Patel, J. Comput. Phys. 88(2), 305 (1990).
- 9. F. Stern, E. G. Paterson, and Y. Tahara, IIHR Report, IIHR, University of Iowa, Iowa City, Iowa, 1996 (in preparation).
- F. Stern, H. T. Kim, D. H. Zhang, Y. Toda, J. Kerwin, and S. Jessup, J. Ship Res. 38(2), 137 (1994).
- 11. F. Stern, J. Longo, and Z. J. Zhang, "Detailed Bow-Flow Data and CFD of a Series 60 CB = .6 Ship Model for Froude Number .316," in *Proceedings*, 24th ATTC, College Station, Texas, November 1995 (unpublished).
- 12. L. Larsson, SSPA Publication No. 90, 1981.
- L. Larsson, V. C. Patel, and G. Dyne, Flowtech International Report No. 2, June 1991.
- 14. K. J. Bai and J. H. McCarthy, *Proceedings, Workshop on Ship Wave Resistance Computations* (DTNSRDC, Bethesda, MD, 1979).
- F. Noblesse and J. H. McCarthy, Proceedings, Second Workshop on Ship Wave Resistance Computations (DTNSRDC, Bethesda, MD, 1983).
- W. T. Lindenmuth, T. J. Ratcliffe, and A. M. Reed, Technical Report DTRC/SHD-1260-01, David Taylor Research Center, 1988 (unpublished).
- H. C. Raven, "Nonlinear Ship Wave Calculations Using RAPID Method," in Proceedings, Sixth International Conf. on Numerical Ship Hydrodynamics, Iowa City, Iowa, August 1993, p. 95.
- J. E. Choi and F. Stern, "Solid–Fluid Juncture Boundary Layer and Wake with Waves," in *Proceedings, Sixth International Conf. on Nu*merical Ship Hydrodynamics, Iowa City, Iowa, August 1993, p. 215.
- Z. Zhang and F. Stern, "Wave-Induced Separation," in ASME Forum on Advances in Numerical Modeling of Free Surface and Interface Fluid Dynamics, San Francisco, CA, 12–17 November 1995 (unpublished).
- E. G. Paterson and M. C. Hyman, F. Stern, P. A. Carrica, F. Bonneto, and D. Drew, "Near- and Far-Field CFD for Naval Combatants Including Thermal Stratification and Two-Fluid Modeling," in 21st ONR Symposium on Naval Hydrodynamics, Trondheim, Norway, June 1996 (unpublished).

2次元モデルによるトランサム船尾近傍の自由表面流場に関する研究*1

正会員田原裕介*2,正会員岩崎泰典*3

A Study of Transom-Stern Free-Surface Flows by 2-D Computational and Experimental Models

By Yusuke TAHARA (Member) and Yasunori IWASAKI (Member)

This paper presents a study of free-surface flow near the transom stern by the two-dimensional experimental and computational models. The experiments were performed at towing tank with main emphasis placed on free-surface and pressure measurements. The computational approach is based on Reynolds-averaged Navier-Stokes equations, and computational grids are fitted to body as well as free surfaces. Nonlinear free-surface boundary conditions are satisfied at the exact location of free surface, which is obtained as part of solution. The present computational model approximately includes breaking wave effects, such that the breaking wave is expressed as the reversed flow region under the free surface which is assumed continuous. Numerical results are presented for three transom sterns, including detailed comparison with experimental data. Discussions are made regarding influences of geometrical differences of transom stern and Froude and Reynolds numbers on the free surface flows as well as frictional and pressure drags acting on the body surface. Satisfactory agreements are demonstrated between the computations and measurements, and the present two-dimensional models appears to be capable to analyze hydrodynamic factors of generation of breaking wave near the transom stern, which are found to be related to boundary layer flow near the stern associated with wave-induced pressure gradients.

Keywords : Transom Stern, Free-surface Flow, Two-dimensional Model, RaNS Equations, Nonlinear Free-surface Conditions

1. 緒言

近年多くの高速痩型船ではトランサム船尾が用いら れている.トランサム船尾は大きなTKM(キールから の横メタセンター高さ)すなわち大きな水線面積を確 保する設計上の利点がある反面,ある限界を超えると 船尾端後方の水面を乱し,大きな船体抵抗増加の原因 となる¹⁾²⁾³⁾⁴⁾.このためトランサム船尾近傍の自由表 面流場を解析し,流場の変化に関与する要因を見い出 すことは船型設計上極めて重要である.岩崎ら⁵⁾はト ランサム船尾近傍の自由表面流場を,(A)船尾端直後 が死水領域となっている状態,(B)船尾端直後が平滑 水面となるが後方で波崩れが生じている状態,(C)船 尾端直後が平滑水面となり後方の波崩れもない状態, の3種に分類した.本論文は,この岩崎らの研究⁵⁾を 補足するものであり,その研究の中で用いた2次元ト ランサム船尾近傍の自由表面流場計算法とその計算結 果について詳細に論じることを目的とする.

トランサム船尾近傍の自由表面流場を計算する手法 はこれまでにも幾つか報告されているが、それらは 非粘性流法と粘性流法に大別できる.非粘性流法の 例として、Reed and Telste⁶⁾ や Telste and Reed⁷⁾は Rankine source と Havelock source を用い、線形自由 表面条件に基づく境界値問題解法を用いてこの問題を 研究している.しかし対象とされているのは高速艇で あり、また船尾端における境界条件の取り扱いと自由表 面の計算精度に重点がおかれ、さらに上述の状態(A) や(B)を考慮するには至っていない.また Raven⁸⁾は パネル法を用い、非線形自由表面条件に基づく計算 コードを開発しており、その一つの機能としてトラン サム船尾を持つ船体にも対応できるようになっている.

^{*1} 平成8年11月15日造船三学会秋季連合大会において 講演,原稿受付 平成8年12月1日

^{*2} 大阪府立大学工学部

^{*3} 川崎重工業

Raven⁸⁾の方法では予め推測された船尾端直後の自由 表面波高から繰り返し計算を開始し,船尾端における 境界条件を満足するように徐々に補正していくもので あり,状態(C)に関する限り妥当な結果を得ることが できる.しかしTelsteら⁶⁾⁷⁾やRaven⁸⁾のように境界値 問題としてこの問題を解く限り船尾端の特異性によっ て生じる困難を解決する必要があり,このため船尾端 の圧力条件や船尾端直後の自由表面の傾斜を仮定する 等が必要となる.またTelsteら⁶⁾⁷⁾やRaven⁸⁾が用いる 自由表面境界条件では,船尾端直後の死水領域や状態 (B)の波崩れ状態を正確に考慮することが難しい.

粘性流法の一つとしては川村ら⁹⁾が開発したレイノ ルズ平均ナビエ・ストークス (RaNS) 方程式に基づく 解法がある.この手法では自由表面に適合していない 計算格子と密度関数法が自由表面境界条件に用いられ, 砕波現象やトランサム船尾近傍で生じる死水現象,す なわち上述の状態(A)にも対応できるようになってい る.川村ら⁹⁾はこれを高速滑走艇に適用し,実験値と の比較を通して手法の有用性を示した.しかし計算格 子が自由表面に適合していないため,自由表面近傍の 詳細な情報が失われる恐れがあり,また川村ら⁹⁾の研 究は密度関数法の適用に重点が置かれていたことも あって,速度変化によるトランサム船尾近傍の自由表 面状態の変化や,上述の(A)(B)(C)の発生限界を検討 するような問題には適用されていない.

本研究では RaNS 方程式に基づいた簡便な 2 次元計 算モデルによる計算手法を用いている. その理由は、本 研究で対象としている最近の痩型船のトランサム船尾 近傍では、岩崎ら⁵⁾の観察により2次元的な流場が重 要で,2次元計算モデルを用いても有益な情報が得ら れると判断したためである.また自由表面近傍の詳細 な流場情報を得るために計算格子は船体および自由表 面に適合しており、計算のタイムステップ毎に更新し ている. さらに本計算モデルでは自由表面は船尾端に 連続するという仮定を用いるために,基本的に上述の 自由表面状態(B)と(C)を計算目的としたものである. 船尾端における条件は圧力条件のみであり、非粘性流 法で必然的に用いられるような船尾端直後の自由表面 傾斜を仮定する必要はない. 非線形自由表面条件は実 際の自由表面位置において与えられ、船尾後方の波崩 れ現象も連続した自由表面という仮定のもとではある が近似的に表現できる計算モデルを用いる.以下に本 計算手法の詳細を述べ,計算結果を3種類の2次元ト ランサム船尾形状について示す. さらに船尾没水量や フルード 数変化による自由表面流場の変化と、摩擦抵 抗/圧力抵抗の変化との関連についても議論する. 岩 崎⁵⁾らによれば、本計算法による結果は状態 (B)(C) に 関して実験結果と良好な一致を示しており、また船尾 直後の自由表面傾斜角を用いることにより(A)(B)(C)の状態がある程度予測可能であることが示されている.

2. 基礎方程式および数値計算方法

2次元非定常/非圧縮流体に関する RaNS 方程式 は、デカルト座標系を用いて以下のように無次元化表 記される.

$$\sum_{i=1}^{2} \frac{\partial U_i}{\partial x^i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \sum_{j=1}^2 \left(U_j \frac{\partial U_i}{\partial x^j} + \frac{\partial \overline{u_i u_j}}{\partial x^j} \right) + \frac{\partial \hat{p}}{\partial x^i} - \frac{1}{\operatorname{Rn}} \nabla^2 U_i = 0$$

with
$$\nabla^2 = \sum_{j=1}^2 \frac{\partial^2}{\partial x^j \partial x^j}$$
 (2)

ここで $U_i = (U, V)$ および $u_i = (u, v)$ は代表速度 U_0 で無次 元化された平均および変動速度成分, $x_i = (X, Y)$ は代表 長 Lで無次元化された座標, $\hat{p} = (p + \rho g Y)/\rho U_0^2 la \rho U_0^2$ で無次元化された Piezometric pressure, $Rn = U_0 L/\nu$ はレイノルズ数, $-\overline{u_i u_j}$ は U_0^2 で無次元化されたレイノ ルズ応力成分である.本研究では2次元問題を対象と ているために,簡便な Baldwin-Lomax 乱流モデルを 用いて $-\overline{u_i u_j}$ を与える. Baldwin-Lomax モデルはタン カー船型等の3次元問題においてはその難点を指摘さ れているが,2次元問題においては妥当な結果を与え ることが報告されている¹⁰⁾.(1)(2)式は数値的に生成 された非直交曲線座標系に変換され,以下のような一 般式を得る¹¹⁾.

$$\frac{1}{J}\sum_{i=1}^{2}\sum_{j=1}^{2}\frac{\partial}{\partial\xi^{i}}(b_{i}^{j}U_{j})=0$$
(3)

$$\sum_{i=1}^{2} \sum_{j=1}^{2} g^{ij} \frac{\partial^2 \phi}{\partial \xi^i \partial \xi^j} - 2 \sum_{j=1}^{2} a^j_{\phi} \frac{\partial \phi}{\partial \xi^j} = R_{\phi} \frac{\partial \phi}{\partial \tau} + S_{\phi} \quad (4)$$

さらに輸送方程式(4)は有限解析法により離散化され, 最終的には次に示される10点離散化方程式となる¹¹⁾.

$$\phi_P^n = \frac{1}{1 + C_P \frac{R}{\Delta \tau}} [C_{NE} \phi_{NE}^n + C_{NW} \phi_{NW}^n + C_{SE} \phi_{SE}^n]$$

 $+C_{SW}\phi_{SW}^{n}+C_{EC}\phi_{EC}^{n}+C_{WC}\phi_{WC}^{n}+C_{NC}\phi_{NC}^{n}$

$$+ C_{SC}\phi_{SC}^{n} + C_P\left(\frac{R}{\Delta\tau}\phi_P^{n-1} - (S_{\phi})_P\right)] \qquad (5)$$

$$-193 -$$

また速度場/圧力場結合には PISO タイプ/ワンステッ プ法を採用する¹²⁾.有限解析法および PISO タイプ /ワンステップ法はこれまでにも 3 次元問題における 実績があり (例えば田原ら¹²⁾⁻¹⁶⁾¹⁹⁾)多様な計算例を 通してその精度が評価・検討されている.基礎方程式 は全て非定常で定義されているが,本研究では定常解 の求解を目的としているために時間 t を収束パラメー タとして用いることになる.また計算格子は物体およ び自由表面 $Y = \eta(X)$ に適合しており,時間ステップ毎 に更新される.



Fig. 1 Solution domain and boundaries





Fig.1 に示すような,一様流 $\vec{U}_0 = (U_0, 0)$ 中に固定さ れた2次元船体を考える.Xを船の長さ方向に,Yを 鉛直上向きにとり,また計算座標系(ξ^1,ξ^2)も各々同方 向にとる.Fig.1には計算領域も示されているが,そ の領域を囲む境界は船体表面Sb,流入境界Si,流出 境界Se,外部境界So,そして自由表面Swである.そ れらの境界で満たされる境界条件はつぎのようにな る.流入条件としてSiにおいて $U=U_0, V=0$;流出条 件として速度場のゼロ拡散と圧力場のゼロ勾配を仮定 して,Seで $\partial^2(U,V)/\partial X^2 = \partial \hat{p}/\partial X = 0$;物体表面の条 件としてSbで $U=V=\partial \hat{p}/\partial n=0$,ここでnは壁面法線 方向;そして自由表面Swでは後述する仮定に従って $\partial(U,V)/\partial Y=0, \hat{p}=\eta/Fn^2$ ($Fn=U_0/\sqrt{Lg}$:フルード 数)である.自由表面は後述するように自由表面運動 学的条件式を解いて求められるが、その場合 $\eta(X)$ が満たすべき条件は船尾端において $\eta=-I_T$ (I_T は船尾没水量), さらに流出境界 Se において $\partial \eta/\partial X=0$ である.



Fig. 3 Two-dimensional model ship

自由表面においては運動学的条件と力学的条件の2 つの条件がある.ここで自由表面は船尾端から繋がり, 船尾端における圧力は自由表面と同じであるという仮 定をする.また船尾後方の波崩れは,連続した自由表 面下の逆流現象で表わされるものと解釈する.この自 由表面モデルを用いれば,本研究で対象とする自由表 面流れを表現する厳密な運動学的条件は従来用いられ ているものと同一になりつぎのように表わされる.

$$\frac{DF}{Dt} = 0 \quad \text{with} \quad F = \eta - Y \tag{6}$$

また厳密な力学的条件は

$$\tau_{ij}n_j = \tau_{ij}^* n_j \tag{7}$$

ここで n は自由表面における単位法線ベクトル, τ_{ij} と τ_{ij}^* は各々流体の内部側および外部側の応力テンソルである.本研究では田原ら¹³⁾⁻¹⁶⁾が用いた近似を(6)(7) 式に適用する.その近似はつぎのような仮定に基づい ている.(a)外部側の応力および表面張力影響は無視 する.(b)法線方向の粘性応力と法線方向および接線 方向のレイノルズ応力を無視する.そして(c)自由表 面曲率は小さいと仮定し,接線方向応力成分における 法線方向速度の接線方向勾配を無視する.(a)(b)(c)の 仮定に基づき,自由表面上で以下の式を得る.

$$\frac{\partial \eta}{\partial t} + U \frac{\partial \eta}{\partial X} - V = 0 \tag{8}$$

$$\widehat{P} = \frac{\eta}{Fn^2} \tag{9}$$

$$\frac{\partial U}{\partial Y} = \frac{\partial V}{\partial Y} = 0 \tag{10}$$

(9)(10)式は圧力場と速度場の境界条件となり,(8)式 は自由表面を求めるために用いられる.ここで(10)式 についてはさらに高次の表現も可能であるが,それは 本研究では考慮しない.(8)式を解いて自由表面が更 新される度,船体および自由表面計算格子が再生成さ れる.その方法は3次元問題における有用性が実証さ れている田原ら¹⁵⁾¹⁶⁾の方法を2次元問題に応用したも のである.(8)式を数値的に解く方法を以下に述べる.



Fig. 4 Convergence history of wave profile (Shi) Model - A, Fn=0.33, Rn=2.920,000)

まず解の収束を加速するために local time stepping を用いる.すなわち,解が定常状態に至った時点で時 間項が消えることを考慮し,計算点における合流速 $|\vec{U}|$ を用いて (8) 式をつぎのように変形する.

$$\frac{\partial \eta}{\partial t} + \frac{U}{|\vec{U}|} \frac{\partial \eta}{\partial X} - \frac{V}{|\vec{U}|} = 0$$
(11)

上式を計算座標系に変換すると下式となる.

$$\frac{\partial \eta}{\partial \tau} + \widetilde{U} \frac{\partial \eta}{\partial \xi^1} - \widetilde{V} = 0 \tag{12}$$

with

$$\widetilde{U} = (U/|\vec{U}| + Xg_{\tau})/X_{\xi^1} \tag{13}$$

$$\widetilde{V} = V/|\vec{U}| \tag{14}$$

ここで Xg_{τ} は格子点の移動速度であるが、後述する 本計算の格子再生成法ではゼロとなる、本研究では (13)(14) 式を解くために空間中心差分法と4 階人工粘 性を用いる、まず (12) 式を以下のように表わす、

$$\eta^{n+1} = \eta^n + \frac{\Delta\tau}{2} [\eta^n_\tau + \eta^{n+1}_\tau]$$
(15)

もしくは

$$\eta^{n+1} = \eta^n - \frac{\Delta \tau}{2} [(\widetilde{U}\eta_{\xi^1} - \widetilde{V})^n + (\widetilde{U}\eta_{\xi^1} - \widetilde{V})^{n+1}]$$
 (16)
これを以下のように近似する。

$$\eta^{n+1} = \eta^n - \frac{\Delta\tau}{2} [\widetilde{U}^n (\eta^n_{\xi^1} + \eta^{n+1}_{\xi^1}) - 2\widetilde{V}^n] \qquad (17)$$

さらに delta form すなわち $\Delta \eta^n = \eta^{n+1} - \eta^n$ と差分演 算子 *L* を用いると下式となる.

$$L[\Delta \eta^n] = \Delta \tau (\widetilde{V} - \widetilde{U}\eta_{\xi^1})^n \tag{18}$$

ここで

$$L = \left[1 + \frac{\Delta \tau}{2} \widetilde{U}^n \frac{\partial}{\partial \xi^1}\right] \tag{19}$$

このとき L 中の空間微分には 2 次中心差分を用いる が,解法の安定化のために陰的に 4 階人工粘性を加え る.さらに陽的にも 4 階人工粘性を加えて解法の安定 度を高め,最終的には以下の差分方程式を解く.

$$L^*[\Delta\eta^n] = \Delta\tau [\widetilde{V} - \widetilde{U}\frac{\partial\eta}{\partial\xi^1} - \epsilon_{\xi^1}\frac{\partial^4\eta}{\partial\xi^{1^4}}]^n \qquad (20)$$

ここで

$$L^* = \left[1 + \frac{\Delta\tau}{2} \left(\widetilde{U}^n \frac{\partial}{\partial\xi^1} + \epsilon_{\xi^1} \frac{\partial^4}{\partial\xi^{1^4}}\right)\right]$$
(21)

(20) 式は5重対角行列ソルバーを用いて簡単に解くこ とができる. (20)(21) 式中の ϵ_{ξ^1} は微小な値でよく,田 原ら¹⁵⁾¹⁶⁾が検討した値を基に,本計算で用いる値を決 定した.類似した方法を池畑ら¹⁷⁾や Jameson ら¹⁸⁾も 用いているが,方程式の導出概念に違いが見られる.

また田原ら¹⁵⁾¹⁶⁾は粘性流場計算格子と自由表面 計算格子を独立に生成し、お互いの情報をbi-linear interpolationを用いて交換することにより安定した解 が高い精度で求め得ることを示している.本計算法に おいても、その手法を利用しているが、数値実験の結 果、船尾後方の波崩れを連続した自由表面下の逆流域 で表現する本研究の自由表面モデルを用いる際に,こ の手法は有効に機能することを確認した.本計算では 60点の粘性流場計算格子に対し,900点の等間隔自由 表面計算格子を用いている.自由表面計算格子の密度 は数値実験によって決定し,この点数以上では格子依 存性が無視できることを確認した.



Fig. 5 Computed wave profiles for Figt-Figte Model and Ship Model - A (Fn=0.3, Rn=2,000,000)

すでに述べたように、計算格子は物体表面および自 由表面に適合している. 粘性流場計算格子の分布は船 の長さ方向に120,上下方向に30である.今回の計算 は船尾端後方の自由表面流れに重点をおいており、計 算格子の密度は田原ら¹⁵⁾¹⁶⁾によって行った数値実験の 結果に基づいて決定した.ここで $\xi^1 = \xi^1(X)$ とし、自 由表面が更新される度に一定 ξ^1 に沿った上下方向の格 子再配置を行う.したがって先に述べた格子点の移動 速度 $X_{g_{\tau}}$ はゼロとなる.本研究の対象は乱流場である ので、物体近傍の法線方向最小格子間隔は田原¹⁹⁾の結 果に従い $Y^+ < 2$ とした. Fig.2 に船尾端近傍の計算格 子を示す.

Fig.1 に示しているように、計算で用いる船体形状 は船体前半で平板形状となり、FPはSiと一致する. すなわち平板乱流境界層の発達はFP位置から開始す ることとする.この計算モデルを用いることで流入境 界条件を簡略化しているが,同様な方法でタンカー船 型周りの流場を計算した結果を見る限り¹⁰⁾,船体前半 部を平板近似したことの船尾端近傍流場への影響は少 ないと考えられる.Fig.1における Soの位置は静止水 面下1船長,また Se は船尾後方1船長の位置にある.



Fig. 6 Comparison of computed pressure and shear-stress profiles for Flat-Plate Model (Fn=0.3, Rn=2,000,000)

2次元船型および実験値

本研究で用いた 2 次元船型を Fig.3 に示す. Ship Model-A と-B の違いは船尾近傍にあり,船尾端の接線 が静水面となす角が各々0°と 10°である.計算におい ては Flat-Plate Model も用いているが,これは船尾端 から FP までが同一喫水とした計算モデルである.本 研究では岩崎ら⁵⁾が行った船尾端後方の波高計測デー タを比較のために用いる.計測は曳航水槽で 2m 模型 を用い,船尾没水量 $(I_T) = 0 \sim 40$ mm, $Fn = 0.24 \sim 0.36$ の範囲で行われた.実験の詳細については岩崎ら⁵⁾を 参照されたい.

4. 計算決結果および考察

4.1 計算条件

本研究で用いた計算条件は: $U_0=1, L(船長)=1,$ $Fn=0.27\sim0.33$ である.計算 $Rn=2,000,000\sim2,920,000$ は実験状態に合わせて決定した.また船尾没水量は $I_T=0.005\sim0.02$ (実験における 2m 模型の 10~40mm に 相当)とした.計算における無次元時間ステップは 0.01, 圧力場の緩和係数は 0.1,速度場については緩和係数を 用いていない.解の収束条件は速度場および圧力場の 変化量が 0.01%以下であるとした.



Fig. 7 Comparison of computed pressure and pressure-gradient profiles for Flat-Plate Model (Fn=0.3, Rn=2,000,000)

Fig.4 は Ship Model-A に関する自由表面計算値の収 束状態を示している.これは岩崎ら⁵⁾が定義する自由 表面状態(C)にあたる.繰り返し計算52,000回から 60,000回の間で解はほとんど変化しない.それ以外の 計算でも状態(B)(C)に関する限り同様であり,一般 に50,000回以内の繰り返し計算で収束解が得られて いる.しかし実験において明らかに状態(A)が確認さ れるケースを計算した場合には,後述する自由表面 下の逆流域近傍で解が微小振動する場合があること が確認された.これは今回の計算モデルでは状態(A) を完全に表現できないことに関係すると考えられる. しかし計算で得られる自由表面傾斜角を用いて状態 (A)の発生を予測することが可能であり、これについ ては後で詳しく述べる.計算に要する時間は、ワー クステーション DEC AlphaStation 200/4/233 を用い、 50,000 回の繰り返し計算で約8時間である.



Fig. 8 Comparison of computed resistance coefficients



4.2 船尾没水量の影響(Fn および Rn を固定し I_T を変化させた場合)

Fig.5 は船速を固定し (Fn=0.3, Rn=2,000,000) 船尾 没水量 (I_T) のみを変化させ, Flat-Plate Model と Ship Model-A について計算した船尾波高である.ここで X=0.0は AP である. $I_T=0.020$ における Flat-Plate Model の場合には, 図中Nで示される自由表面勾配 の不連続点が現われる.これは後に詳しく述べる通り N 近傍の自由表面下に逆流域が存在し,本計算モデ ルでは波崩れを表現している.この自由表面勾配の 不連続点は $I_T=0.05$ と $I_T=0.010$ の場合には見られ ず,後方まで滑らかに繋がった自由表面となっている. Ship Model -A に関しては全ての結果に勾配の不連続 点が見られる.また I_T が増加するに従い, N の位置 が船尾に近づく傾向が見られる.





Fig.6 は Flat-Plate Model 上の圧力分布 ($Cp = \hat{p}/0.5$) および摩擦応力分布 ($C_{\tau} = \tau_w/0.5\rho U_0^2$)を示している. ここで示す圧力は無次元化された piezometric pressure であり自由表面上では $2\eta/Fn^2$ の値となる. I_T が増加す るに従い船尾近傍の圧力は低くなり, 摩擦応力は高く なる傾向を示す. Fig.7 は AP 近傍における Flat-Plate Model 上の圧力分布および圧力勾配分布である. ここ で AP 以後の圧力分布は自由表面上のものであり, そ の勾配は自由表面勾配を表わしている. 圧力分布の N 点が示す勾配の不連続点は圧力勾配分布の N 点に対応 する. I_T が増加するに従い船尾端における圧力は低下 し、前方から船尾端にかけてより低い負の圧力勾配 が見られる.船尾端以後で圧力勾配は急変して正に 転じるが、 I_T が大きい程高い正の圧力勾配が見られ、 $I_T=0.020$ の場合N点において最大となる.



Fig. 11 Measured and computed wave profiles for Ship Model - A $(I_T=0.01)$

Fig.8 は表面上の圧力分布および摩擦応力分布を 積分して得られた抵抗値を示す. 図中 CPは圧力抵 抗成分を, CFは摩擦抵抗成分を表わす. Flat-Plate Model の場合, 当然ながら圧力抵抗成分はゼロとなる. Flat-Plate Model および Ship Model-A の各ケースに おいて I_T が増加するに従い摩擦抵抗成分が増加する. この傾向は Fig.6 で示された摩擦応力分布に対応して いる. Ship Model-A に関しては, I_T が増加するに従 い圧力抵抗成分が増加している.

まず Flat-Plate Model について計算結果を考察す る.自由表面勾配の不連続点で表わされる自由表面下 の逆流域,すなわち本計算モデルによる波崩れの発生 に関与する要因として,船尾端後方における圧力勾配 を考えてみる.船尾没水量が大きくなるに従い,船尾 端における圧力が低下するとともにその直前に至る物 体近傍流域の負の圧力勾配も低下する.それによって 流体は加速され,境界層が薄くなり,摩擦応力が増す



Fig. 12 Comparison of solutions near stern for Ship Model - A $(I_T=0.005)$

-199-



Fig. 13 Comparison of solutions in reversed flow region near free surface for Ship Model - A ($I_T=0.005$, Fn=0.27, Rn=2,390,000)

ことにより摩擦抵抗係数も増加する.船尾端以後,自 由表面は静水面のレベルまで上昇しようとするが,船 尾没水量が大きいほどその勾配が大きい.それによっ て流場にはより大きい正の圧力勾配が存在し,これは 流体を減速させる効果を持つ.自由表面勾配が流体の 加速や減速に関与していることは,船体周りの自由表 面流場および境界層/伴流場に関する実験的研究²⁰⁾²¹⁾ や計算的研究¹³⁾⁻¹⁶⁾でも示されている. $I_T=0.020$ の 場合,その圧力勾配が流体を逆流させるまでに大きく, よって実際の現象としては波崩れという形になったも のと考えられる.



Fig. 14 Comparison of computed resistance coefficients for Ship Model - A

同一の Fn および Rn における Ship Model-A の計 算結果では、全ての I_T で自由表面下の逆流現象が見ら れていることにより、その現象の有無は船尾端近傍の 境界層の状態にも関係があると考えられる。船尾端近 傍の境界層は Ship Model-A の方が Flat-Plate Model よりも厚く、壁面近傍の流体も減速されており、船尾 端から流出した後自由表面近傍で逆流しやすい状態に なっていたと考えられる。船尾没水量が大きいほど逆 流域が船尾に接近しており、それにともなって逆流域 の規模が大きくなったため、より大きなエネルギー損 失がおこり、圧力抵抗成分が増加する傾向を示すもの と考えられる.逆流域におけるエネルギー損失につい ては、後に Ship Model-A に関する粘性流場の計算結 果を検討する際により詳しく述べる.

4.3 船速の影響(*I*_Tを固定し *Fn* および *Rn* を変化 させた場合)

ここでは船速の影響を調べるために船尾没水量 (*I*_T) を固定し、Fn および Rn を変化させた場合を考え る. Fig.9 は Ship Model-A に関し, $I_T=0.005$ および $I_T=0.010$ において Fn と Rnを変化させて計算した船 尾波形状を示している. I_T=0.005の場合,図中Nで 示す自由表面勾配の不連続点は Fn=0.27 および 0.30 で現われる.また Fn=0.27のNの方がより船尾に近 い. I_T=0.010 の場合は設定した全ての Fn で自由表面 勾配の不連続点が見られ, Fn が小さい程 N 点は船尾 に近くなる. それらの計算値を Fig.10 および Fig.11 において実験値と比較する. IT=0.005/Fn=0.33の場 合,実験で観察された自由表面状態は岩崎ら⁵⁾が定義 する状態(C)にあたり、計算値と実験値は良い一致を 示している. I_T=0.005/Fn=0.27 および 0.30 の場合 は状態(B)が観測されているが、図中N点の位置は波 崩れが船尾方向に向けて始まる位置とほぼ一致して いる.波崩れによるバブル状の領域を除き,計算値 と実験値の一致は良い. IT=0.010/Fn=0.33 および 0.32 の場合は状態 (B) になることが実験で観測され ているが、この場合もバブル状領域を除いて計算値

Computation	Ship model	Stern condition		Froude num. Wave slope		Stern wave condition	
Case No.		Angle(deg.)	Ι _τ	Fn	angle(deg.)	Cal.	Exp.
Case 7	A	0	0.005	0.330	3.4440	Perf.smooth	Perf.smooth
Case 8	A	0	0.005	0.300	4.2768	Smooth	Smooth
Case 9	A	0	0.005	0.270	5.7240	Smooth	Smooth
Case 10	А	0	0.010	0.330	4.5690	Smooth	Smooth
Case 11	А	0	0.010	0.320	4.7711	Smooth	Smooth
Case 12	A	0	0.010	0.300	5.8919	Dead water	Dead water
Case 13	A	0	0.010	0.270	8.2298	Dead water	Dead water
Case 14	A	0	0.020	0.300	10.5077	Dead water	Dead water
Case 15	В	10	0.005	0.300	17.2020	Dead water	Dead water
Case 16	В	10	0.010	0.300	35.0557	Dead water	Dead water
Case 17	В	10	0.020	0.300	63.7170	Dead water	Dead water

Table 1 Relation between observed water-surface condition and computed free-surface angle at stern

は実験値と良い一致を見せている.しかしながら実 験で状態(A)が観測される $I_T=0.010/Fn=0.30$ および 0.27の場合は,船尾端直後の計算値と実験値の間に差 がみられる.実験によれば $I_T=0.010/Fn=0.30$ は状態 (B)から状態(A)に移行する限界付近にあたり,図中 N点の位置でやはり波崩れの開始が確認され,その近 傍における計算値は実験値と良い一致を見せている.

Fig.12 は Ship Model-A の I_T=0.005 における粘性 流場の圧力(Cp),圧力勾配,速度成分(U,V),そし τ Total Head $H = \sqrt{Cp + U^2 + V^2} \mathcal{O} \exists \mathcal{V} \mathcal{P} - \mathcal{C} \mathcal{B} \mathcal{A}$. Fnが大きくなる程,船尾端における圧力は高くなり, また船尾端直後の自由表面近傍における X方向圧力勾 配は小さくなる. 前述のように Fn=0.27 と 0.30 に おいては自由表面勾配の不連続点が見られるが、その 近傍にはUの負の値が見られ、すなわち流れが逆流し ていることを示している.また負のUが表われる近傍 には負のVも認められる. Fn=0.27 と 0.30 における 逆流域近傍で Hが低くなっている場所が見られるが, これは逆流現象によるエネルギー損失を表わしている. I_T=0.005, Fn=0.27の解について, 逆流域近傍を拡大 したものが Fig.13 であり、速度成分 (U,V) コンター、 速度ベクトル (\vec{U}) , それに基づく流線, 渦度 (ω) コ ンター、そして Hコンターが表示してある. Fig.12 で 述べた逆流域近傍のUとVの状態がより明確にわかる. また逆流域近傍において渦度が高くなる場所があり, これは実現象では波崩れによって生成される渦度と考 えられる.またその領域近傍でHが小さくなっており, これは逆流によるエネルギー損失があることを示して いる. Fig.14 は Ship Model-A の I_T=0.005 における表 面圧力分布および摩擦応力分布を積分して得られた抵 抗値である. 図中の CFと CPは Fig.8 で用いられたも のと同定義である.速度が増すに従いCFは減少する. しかし*CP*は波崩れが表現されている*Fn*=0.27と0.30 の間で上昇し、波崩れが消える Fn=0.33 では減少して いることが示されている.

つぎに計算結果を考察する.先に Flat-Plate Model で議論したように、船尾後方における波崩れ現象を船 尾端自由表面近傍の圧力と圧力勾配とに関係づけて考 えてみる. 速度(Fn)が増すに従い船尾端直後自由表 面上の圧力は $Cp=2\eta/Fn^2$ の関係より絶対値が小さく なるとともに流体の慣性が大きくなるために自由表 面の勾配はなだらかになり、圧力勾配は小さくなり自 由表面近傍の逆流現象は起きにくくなる. I_T=0.005, Fn=0.27 で生じた自由表面近傍の逆流現象は速度が上 昇するにつれて後方へ移動し,やがて消滅して岩崎ら ⁵⁾が定義する自由表面状態(C)に移行する. Fig.9 で 示すように速度が増すと船尾後方の波高は上昇してい く、Fn=0.27 と 0.30 間での圧力抵抗の上昇はその造 波によるものと考えられるが,より速度が増して自由 表面状態が(C)に移行するときに圧力抵抗が減少して いるのは逆流現象によるエネルギー損失がなくなった ためであろう.これに類似する現象を研究した例とし て、Jeong ら²²⁾は自由表面下における渦の生成を数値 計算で示し、また川村ら⁹⁾は自由表面の砕波によるエ ネルギー損失を同じく数値計算で示している.

つぎに Fig.12 および Fig.13 に示す速度成分につい て考察する.ここで示す計算値は定常状態のものであ り,時間項を除いた自由表面運動学的条件式から自由 表面における上述の Uと Vの符号は物理的に正しい. また自由表面の頂上 ($\partial \eta / \partial X = 0$)における Vはゼロ になっているが,これは Uがゼロでない値を示してい る領域であることから自由表面運動学的条件式に合致 する.本計算モデルは波崩れを近似的に表現するが, 状態 (B)(C)における自由表面形状の予測はある程度 可能であると思われる.特に流場の状態が (C)に近く なるにつれ,全体的に実験値と良い一致を見せている. Fig.10 に示されている状態 (B)においては,N点の位 置および速度が上昇するに従いN点が船尾端に接近す

る傾向は実験値と良く一致しており、また船尾端直後 とN点以後で計算値は実験値と一致しているが、その 間のバブル状流域が実験で観測された部分を正確に予 測するには至っていない. これは本計算モデルがバブ ル状になった自由表面を表現できないためである。さ らに Fig.11 に示す明らかに状態 (A) である場合の実験 値との不一致の理由は、本計算モデルにおいて船尾端 から連続する自由表面が仮定されているためである. ここで,状態が(C)→(B)→(A)と移行するに従っ て,船尾直後の自由表面傾斜が大きくなるという傾 向に着目し,自由表面傾斜角を用いた自由表面状態 の予測を試みた. Table 1 には本研究で対象とされた 船型, すなわち Ship Model-A および-B について, 船 尾没水量 (I_T) と船速 (Fn および Rn) を変化させた 場合の,船尾端直後の自由表面が静止水面となす角 が示してある.また自由表面の状態が(A)であるか, (B) もしくは(C) であるかを実験で観測した結果が (状態 A:Dead water; 状態 B:Smooth; 状態 C:Perfect Smooth) で示してある. 計算値を用いてそれらの発生 限界を検討したところ、傾斜角約3.5°と5.5°付近に自 由表面状態の変化点があることがわかった. すなわち 約 3.5°以下は状態 (C) であり,それより大きく約 5.5° 以下は状態(B)であり、それを超えると状態(A)であ るとして, その判断基準のもとに自由表面状態を予測 した結果も Table 1 に示す. この結果は実験の観測結 果と良く一致しており、本計算法による結果を用いて 自由表面状態を大まかに予測できる可能性を示してい る. 先に自由表面状態が変化する要因として圧力勾配 を取り上げたが、自由表面傾斜はその近傍流場の圧 力勾配に直接関与することを考えれば妥当な指標と 思われる.ちなみに、本計算および実験で考慮した 計算条件に関する限り, Ship Model-B は全て状態(A) であるという結果になっている. Flat-Plate Model と Ship Model-A を比較した結果から考察すると、 Ship Model-BにはShip Model-Aのように船尾近傍の凹部 がなく、境界層はより厚くなっており、壁近傍の速度 はより減速され, 逆圧力勾配によって船尾後方の流れ が逆流しやすい状態になっていたためと思われる.

5. 結言

本研究では、2次元 RaNS 方程式に基づく数値計算 法を用いて高速痩型船トランサム船尾近傍の自由表面 流場を解析した.計算には物体/自由表面適合格子を 用い,非線形自由表面条件を実際の自由表面位置にお いて与え,船尾後方の波崩れ現象を近似的に表現でき る計算モデルを用いた.自由表面は船尾端に連続する という仮定を用い,岩崎ら⁵)が定義する自由表面状態 (B:船尾端直後が平滑水面となるが後方で波崩れが生 じている状態)と(C:船尾端直後が平滑水面となり後 方の波崩れもない状態)を目的とする計算を行った. 本研究の結論をまとめると以下の通りである.

(1)自由表面状態(B)における船尾後方の波崩れ 現象を,連続した自由表面下の逆流で表わすという計 算モデルを用いた結果,船尾端直後および波崩れの開 始点後方で実験値と良く一致する波高が得られた.ま た波崩れの開始点についても実験による観測と良く一 致した.しかしながら,波崩れの開始点前方船体側の 流場がバブル状になっている領域を正確に表現するに は至らなかった.自由表面状態(C)の波高に関しては, 全体的に実験値と良く一致する結果を得た.

(2)速度を固定して船尾没水量を変化させた場合, 船尾没水量が増えるに従い摩擦抵抗係数および圧力抵 抗係数が増える傾向があることが分かった.また船尾 没水量が増えるに従い自由表面状態が変化すること が分かった.今回設定した速度において,Flat-Plate Modelの場合は状態(C)から(B)へ移行し,さらに Flat-Plate Model および Ship Model-A で状態(B)の 場合,船尾没水量が増えるに従い逆流域が船尾に接近 する傾向が認められた.

(3) 船尾没水量を固定して速度を変化させた場合, 速度が増えるに従い波崩れの開始点は後方へ移動し, やがて逆流域(実験における波崩れ)は消えて状態 (C)に移行する傾向が計算および実験で示された.こ の場合速度が増えるに従い摩擦抵抗係数は一様に減少 し,自由表面状態が(B)である速度域では圧力抵抗係 数は増すが,さらに速度が増して状態(C)に移行する と圧力抵抗係数が減少することが計算で示された.

(4)自由表面状態(B)における逆流域近傍の粘性 流場では渦度の生成が確認され,さらに水頭損失も認 められた.また逆流現象が生じる原因を船尾端近傍の 圧力(piezometric pressure)およびその勾配と関連づ けて議論することを試み,さらに船尾端直後の自由表 面傾斜角を用いて,自由表面状態(A:船尾端直後が 死水領域となっている状態)および状態(B)(C)がある 程度予測できることが示された.

本計算法は,自由表面状態 (B)(C) において妥当な 結果を与えることが示されたが,状態 (A) を厳密に考 慮する計算法への拡張が必要である.このためには物 理空間を船尾端前後で分離し,その分離境界面で整合 条件を満足させる複合計算空間法(最近の例では平田 ²³⁾,田原²⁴⁾等)が有効であると思われる.さらに,よ り一般的な船型に対応するためには3次元性を考慮し た計算法に拡張する必要がある.これは今回の研究で 開発された技術を,田原ら¹⁵⁾¹⁶⁾による3次元船体周り の自由表面流場計算法へ導入することで可能になると 思われる.

最後に、本研究を行うにあたり、種々のご助言ご協 力をいただきました大阪府立大学工学部海洋システム 工学科姫野洋司教授,奥野武俊教授に感謝の意を表わ します.

参考文献

- Yamano, T., Saito, Y., Iwasaki, Y. and Funeno, I.: A Consideration on Stem Form for Fine Ships, 関西造船協会誌, 第 225 号, 1996, pp. 25-35.
- 2) Yamanno, T., Saito, Y., Iwasaki, Y., Taguchi, K. and Maeda, N.: Development of a New Stern Fom for Ocean Going Fine Ships, 関西造船協会誌, 第 221 号, 1994, pp. 25-33.
- 3) 岩崎泰典,山野惟夫: 痩型船型の新形式船尾形状の 開発-復元性能と推進性能-,関西造船協会誌,第 225 号,1996, pp. 7-13.
- 4) 岩崎泰典,山野惟夫: 痩型船型の新形式船尾形状の 開発-耐航性能と操縦性能-,関西造船協会誌,第 226号, 1996, pp. 1-6.
- 5) 岩崎泰典,田原裕介,奥野武俊,姫野洋司,山野惟 夫: 痩型船型の船尾端形状と船尾没水位に関する研 究,日本造船学会論文集,第180号掲載予定,1996.
- 6) Reed, A., Telste, J. and Scragg, C.: Analysis of Transom Stern Flows, Proc. 18th ONR Symposium on Naval Hydrodynamics, Ann Arbor, Michigan, 1990, pp. 207-219.
- Telste, J. and Reed, A.: Calculation of Transom Stern Flows, Proc. 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, Iowa, 1993, pp. 79-92.
- Raven, H.C.: Nonlinear Ship wave Calculations using the RAPID method, Proc. 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, Iowa, 1993, pp. 95-118.
- Kawamura, T. and Miyata, H.: Simulation of Nonlinear Ship Flows by Density-Function Method (Second Report), 日本造船学会論文集, 第 178 号, 1995, pp. 1-7.
- Proc. CFD Workshop Tokyo 1994, Tokyo, Vol. 1, March 1994.
- Chen, C.J. and Chen, H.C.: Finite Analytic Numerical Method for Unsteady Two-Dimensional Navier-Stokes Equations, J. of Computational Physics, Vol. 53, 1984, pp. 209-226.
- 12) Tahara, Y.: Computation of Viscous Flow around Series 60 Model and Comparison with Experiments, 関西造船協会誌, 第 220 号, 1993, pp. 29-47.

- Tahara, Y., Stern, F. and Rosen, B.: An Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number, J. Computational Physics, Vol. 98., No. 1, January 1992, pp. 33-53.
- 14) Tahara Y. and Stern, F.: Validation of an Interactive Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number, J. Computers and Fluids, Vol. 23, No. 6, 1994, pp. 785-816.
- 15) Tahara Y. and Stern, F.: A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number, Proc. CFD Workshop Tokyo 1994, Tokyo, Vol. 1, March 1994, pp. 45-55; also J. Computational Physics, Vol. 127, 1996, pp. 398-411.
- 16) Tahara Y.: Evaluation of a RaNS Equation Method for Calculating Ship Boundary Layers and Wakes Including Wave Effects, 日本造船学会 論文集, 第180 号掲載予定, 1996.
- 17) Ikehata, M., Inoue, T. and Liu, H.: Numerical Simulation of Flow, Waves and Resistances of Series 60 (Cb=0.6) Model, Proc. CFD Workshop Tokyo 1994, Tokyo, Vol. 1, March 1994, pp. 35-44.
- 18) Farmer, J., Martinelli, L., and Jameson, A.: Multigrid Solutions of the Euler and Navier-Stokes Equations for a Series 60 Cb=0.6 Ship Hull for Froude Numbers 0.160, 0.220 and 0.316, Proc. CFD Workshop Tokyo 1994, Tokyo, Vol. 1, March 1994, pp. 56-65.
- 19) Tahara, Y.: An Application of Two-Layer k-ε model to Ship Flow Computation, 日本造船学 会論文集,第177号, 1995, pp. 161-176.
- 20) Toda, Y., Stern, F. and Longo, J.: Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 CB=.6 Ship Model
 Part 1: Froude Numbers .16 and .316, J. of Ship Research, Vol. 37, No. 4, December 1992, pp. 360-377.
- 21) Longo, J., Stern, F. and Toda, Y.: Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 CB=.6 Ship Model -Part 2: Scale Effects on Near-Field Wave Patterns and Comparisons with Inviscid Theory, J. of Ship Research, Vol. 37, No. 1, March 1993, pp. 16-24.
- 22) Jeong, U.C. and Doi, Y.: Numerical Study of Vortical Flows beneath the Free Syrface around

Struts, 日本造船学会論文集, 第178号, 1995, pp. 23-31.

- 23) Hirata, N. and Kodama, Y.: Flow Computation for Three-Dimensional Wing in Ground Effect Using Multi-Bloack Technique, 日本造船学会論文集, 第 177 号, 1995, pp. 49-57.
- 24) Tahara, Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, 関西造船協会誌, 第 226 号, pp. 63-76.

-13----

非H型トポロジー自由表面パネルを用いた Dawson-Type Rankine-Source 法による定常造波問題の数値計算 一第一報:特にO型自由表面パネルの適用について-^{*1}

正会員田原裕介*2

A Numerical Approach for Steady Ship-Wave Problem Based on Dawson-Type Rankine-Source Method with Non-H-Type-Topology Free-Surface Panels

- 1st Report: with Main Emphasis on Application of O-Type-Topology -

By Yusuke TAHARA (Member)

This paper presents a numerical approach for steady ship-wave problem, which is based on extension and modification of Dawson-type Rankine-source method in order to utilize arbitrary topology of free-surface panels. The wave radiation condition is numerically given, where 3rd-order upstream differential operator evaluates 2nd-derivative terms in the free-surface boundary condition. In this 1st report, the main emphasis is placed on application of O-type as well as conventional H-type topology free-surface panels. It appears that the former offers advantages over the latter due to the efficient panel distribution and improved resolution of the local-region bow- and shoulder-wave systems. The computational results are presented for Wigley and Series60 C_B =0.6 ship models including detailed comparisons with experimental data.

Keywords : Steady Ship-Wave Problem, Rankine-Source Method, Arbitrary Topology Panels, Numerical Radiation Condition, Wigley Model, Series60 Ship Model

1. 緒言

Dawson¹⁾以来,定常造波問題の解法としての Rankinesource 法(以下 R-S 法と略称)は多くの研究者によっ て研究された.その間,非線形影響,揚力体問題への 応用,粘性流法とのカップリングなどが提案されて いる.R-S 法は計算労力が RaNS(Reynolds-averaged Navier-Stokes)方程式法に代表される粘性流法より少 なく,妥当な精度で自由表面流場を計算できることが大 きな利点である²⁾.しかし解が自由表面パネルの配置方 法に依存しやすいことや,計算領域外端の位置によっ ては波の反射が起こるなどといった問題点が指摘され, 汎用性のある船型デザインツールとなるためには,ま だ幾つかの解決すべき課題が残されている.計算手法 と自由表面パネルトポロジーの独立性という問題もそ の一つである.本研究では特定の自由表面パネルトポ ロジーに制限されない R-S 法の開発を試み,従来一般 的に用いられてきた H 型以外のトポロジー,本報では 特に O 型トポロジーの有用性を示すことを目的とする.

RaNS 方程式法などの粘性流法では,計算格子のト ポロジーをその全景から大まかに定義する.例えば 格子配列が長方形型のものをH型,同心円型のもの をO型と呼ぶ.一般に数値計算法自体は計算格子(パ ネル)トポロジーと独立し,対象とする物体形状に応 じて適当なトポロジーが選択できなければならない. Dawson¹⁾以来,これまでに開発されてきた定常造波 問題に対する R-S 法では全てH型トポロジーが用い られている.類似した方法(Source-doublet パネル法:

^{*1} 平成9年5月24日 関西造船協会春季講演会において 講演,原稿受付 平成9年6月10日

^{*2} 大阪府立大学工学部

SPLASH code) で Rosen³⁾が O 型を用いているが,自 由表面条件は Dawson¹⁾の提案したものとは異なってお り,またその数値的な取り扱いについて詳細な記述が ない. 江口⁴⁾は H 型トポロジーパネルを変形し,船体 外部領域で階段状のパネル列を用いることを提案した が,基本的なパネルトポロジーを変更するには至って いない.また船体水線面形状に適合した H 型トポロ ジーパネルは Blunt な船首近傍などで大きく捻れ,計 算精度および計算安定性を低下させる.



Fig. 1 Definition sketch of coordinate system



Fig. 2 Indices of computational coordinates on free-surface panels

定常造波問題に対する R-S 法において, H 型自 由表面パネルトポロジーが採用されてきた一因には Radiation 条件の問題があると考えられる.現在一般 的に用いられる方法は,Dawson が提案した上流差分 による方法,およびその後提案された Panel-shift 法 (初期および最近の例として文献⁶⁾⁻⁸⁾など)に大別でき る.Dawson が導いた自由表面条件式は二重模型流れ の流線座標系で記述されており,その座標系をそのま ま用いて上流差分による Radiation 条件を与える場合 には H 型パネルトポロジーが用いやすい.これに対し て Panel-shift 法を用いる場合,その shift する方向が 上流側とされており,基本的に使用できるトポロジー は H 型に限られる.江口⁴⁾の R-S 法も Panel-shift 法を 用いており,パネルトポロジーを変更せず階段状の H 型パネルを採用したのはそのためであると考えられる.

自由表面条件の数値的操作に関し、上流差分法 と Panel-shift 法は同様な意味合いを持つ⁹⁾. それは dissipation を加えることにより数値的に Radiation 条 件を与えることであり、原理的には Panel-shift 法の dissipation は上流差分法に比べ小さく、よって波の後 方減衰が少ない^{4),9)}. しかし有限差分法は適当な座標 変換によって物理空間とは独立に定義された計算空間 で用いることにより、任意の物理空間格子(パネル)ト ポロジーに対応できるという利点がある. Dissipation はパネルサイズにも関係しており、必要な領域で集中 的に密度を上げることができるようなパネルトポロ ジーを用いれば、その影響を必要最小限度に止めるこ とが可能である.

本研究では上述した有限差分型 Radiation 条件の利 点を応用し、Dawson-type R-S 法を任意の自由表面パ ネルトポロジーに対応できるように改良した.物理空 間で定義された自由表面条件式の一部を計算空間に変 換し、その変換座標系で上流差分法を定義することに より,自由表面パネルトポロジーに制限されないR-S 法の適用が可能となった.自由表面条件は荻原ら10)が 提案し、その後著者ら11)も採用した非線形条件式に基 づいているが,本研究ではその第一近似解,すなわち 従来多く用いられている Dawson が提案した線形近似 条件の解を示す. 自由表面パネルには従来の H 型トポ ロジーに加え、船体近傍の解像度重視という観点の もと、その領域で集中的にパネル密度を上げやすく、 加えて Blunt な船首にも対応しやすい O型トポロジー も採用した.また〇型トポロジーパネルの生成には 著者が RaNS 方程式法に関する研究で開発したポア ソン方程式ソルバー¹²⁾を用いた.本研究では,手法 評価の第一段階として Wigley モデルおよび Series60 C_B=0.6 モデルを対象船型とした.これらの船型は過 去の実験および計算例が豊富であり、計算結果の自由 表面場および造波抵抗に関する詳細な検討が可能であ る. Blunt な船首を持つ船型や,水面貫通翼に適用し た結果は続報で報告する.計算結果を検討した結果, O型自由表面パネルはH型に比べより効率的に計算領 域を包括でき, また船首波系や肩波系をより正確に予 測できることが確認された.
2. 基礎方程式および数値計算方法

デカルト座標系を Fig.1 に示すように定義し,一様 $\hat{\pi}\vec{U}_{\infty} = (U_0, 0, 0)$ 中に固定された船体を考える. 流体 を非粘性/非圧縮/非回転であると仮定すれば,速度 場は次のように定義できる.

$$u(x, y, z) = \Phi_x$$
 , $v(x, y, z) = \Phi_y$,
 $w(x, y, z) = \Phi_z$ (1)

ここでu, v, wはx, y, z軸方向の速度,そして $\Phi(x, y, z)$ は 流体中でラプラス方程式 $\nabla^2 \Phi = 0$ を満足する速度ポテン シャルである. $\Phi(x, y, z)$ は船体表面上で $\partial \Phi / \partial n = 0$ (nは表面法線方向)を満たし,また自由表面 $z = \eta(x, y)$ 上で以下に示す運動学的条件と力学的条件を満足する.

 $\Phi_x \eta_x + \Phi_y \eta_y = \Phi_z \qquad (z = \eta) \tag{2}$

$$\frac{1}{2g} [\Phi_x^2 + \Phi_y^2 + \Phi_z^2 - U_0^2] + \eta = 0 \qquad (z = \eta)$$
(3)

上式でgは重力加速度である.ここで $\eta(x, y)$ およびその勾配が小さいと仮定する.そして静止水面について テーラー展開を行い,さらに Φ を2重模型速度ポテン シャル Φ_0 と自由表面流れを表わす速度ポテンシャル Φ_1 の和とし,2つの条件式を合成したのち,2重模型流 れの流線座標に変換すると以下の式を得る¹⁰).

$$\Phi_{0l}^{2}\Phi_{1ll} + 2\Phi_{0l}\Phi_{0ll}\Phi_{1l} + g\Phi_{1z}$$

= $-\Phi_{0l}^{2}\Phi_{0ll} + gD_{1} - \Phi_{0l}D_{2l}$ (z = 0) (4)

ここで

$$D_{1}(x, y) = \Phi_{1x}(\eta_{x} - \eta_{0x}) + \Phi_{1y}(\eta_{y} - \eta_{0y}) - (\Phi_{0zz} + \Phi_{1zz})\eta$$
(5)

$$D_2(x,y) = -\frac{1}{2g} (\Phi_{1x}^2 + \Phi_{1y}^2 + \Phi_{1z}^2)$$
$$-\frac{1}{g} [(\Phi_{0x} + \Phi_{1x})\Phi_{1xz} + (\Phi_{0y} + \Phi_{1y})\Phi_{1yz}]$$

$$+\Phi_{1z}\Phi_{1zz}]\eta\tag{6}$$

$$\eta_0(x,y) = \frac{1}{2g} (U_0^2 - \Phi_{0x}^2 - \Phi_{0y}^2)$$
(7)

上式でlは2重模型流れの流線座標, D_1 および D_2 は η の高次項を省略した Φ_1 に関する非線形項である.以上述べた境界条件のもとに Φ_0 と Φ_1 が決定すれば, p_∞ を無限遠点の圧力としたときの圧力場p,自由表面波高 $\eta(x,y)$,さらに表面単位法線ベクトルを \vec{n} としたときに船体に働く流体力 \vec{F}_b は次式で与えられる.

$$p = p_{\infty} + \frac{1}{2}\rho[(U_0^2 - (\Phi_{0x} + \Phi_{1x})^2)]$$

$$-(\Phi_{0y} + \Phi_{1y})^2 - (\Phi_{0z} + \Phi_{1z})^2]$$
(8)

$$\eta(x,y) = \frac{1}{2g} (U_0^2 - \Phi_{0x}^2 - \Phi_{0y}^2 - 2\Phi_{0x}\Phi_{1x} - 2\Phi_{0y}\Phi_{1y}) - D_2(x,y)$$
(9)

$$\vec{F}_b = -\int_{S_b} (p - p_\infty) \vec{n} dS \tag{10}$$

自由表面境界条件(4) は非線形であり、 $\Phi_0 \ge \Phi_1 \ge 2$ するためには局所線形解を仮定した繰り返し計算など を用いなければならない、その試みは荻原ら¹⁰⁾や著者 ら¹¹⁾が行い、線形解より実験値に近い解を得ている、 しかしながら本研究では D_1 および D_2 をゼロとした第 一近似式、すなわち Dawson¹⁾によって導かれたもの と同一の線形自由表面条件式を用いた求解までに止め る、非線形影響の考慮は今後の研究で行う予定である、



Fig. 3 Panel arrangement on free surface (Wigley model)

一般に開発されている R-S 法と同様,本研究では速 度ポテンシャル Φ_0 と Φ_1 を船体および静止水面上に分布 した Rankine source を用いて表現する.すなわち,流 場の点 P(x, y, z)における Φ_0 と Φ_1 は次式で与えられる.

$$\Phi_0 = \Phi_{\infty} - \int_{S_b} \sigma_{0b} (\frac{1}{r} + \frac{1}{r^*}) dS$$
 (11)

$$\Phi_1 = -\int_{S_b} \sigma_{1b} (\frac{1}{r} + \frac{1}{r^*}) dS - \int_{S_f} \sigma_{1f} (\frac{1}{r}) dS \quad (12)$$

ここで Φ_{∞} は一様流の速度ポテンシャル, σ_{0b} と σ_{1b} は船 体表面 (S_b)上の吹き出し密度, σ_{1f} は静止水面 (S_f)上 の吹き出し密度,rは Pと特異点間の距離,そして r^* は Pと鏡像特異点間の距離である.



Fig. 4 Density distribution of free-surface panels (Wigley model)

次に計算手法について述べる.本研究で用いる手法 の基本部分は R-S 法のスタンダードなものであるので 概略を記すに止める.まず境界面 S_f および S_b を四角 形パネルで分割し,各パネル上で吹き出し密度一定と 近似する.ここで Φ_0 が船体表面上で満たすべき境界条 件を考慮し,積分方程式を離散化すると, S_b 上の*i*-番 パネルの代表点(図心)において次式を得る.

$$2\pi\sigma_{0bi} + \sum_{j=1}^{N_0} A_{0_{ij}}\sigma_{0bj} = -\vec{n}_i \cdot \vec{U}_{\infty} \quad (i=1,N_0) \quad (13)$$

ここで N_0 は S_b 上のパネル数, $A_{0_{ij}}$ は法線方向撹乱速 度マトリクスである. (13) 式より構成されるマトリク ス方程式は船体形状が与えられれば解くことができる. 一方 Φ_1 が船体および自由表面上で満たすべき境界条件 を考慮し,同様な方法で積分方程式を離散化すると, Sb上のi-番パネルの代表点において次式を得る.

$$2\pi\sigma_{1b_i} + \sum_{j=1}^{N_0} A_{0_{ij}}\sigma_{1b_j} = \sum_{j=1}^{N_1} A_{1_{ij}}\sigma_{1f_j} (i=1, N_0) \quad (14)$$

ここで N_1 は S_f 上のパネル数, $A_{1,j}$ は自由表面上に分布した σ_{1f} による船体法線方向撹乱速度マトリクスである.また自由表面 S_f 上の i-番パネルの代表点においては次のようになる.

$$\sum_{j=1}^{N_1} A_{2_{ij}} \sigma_{1f_j} - 2\pi g \sigma_{1f_i}$$
$$= \sum_{j=1}^{N_0} A_{3_{ij}} \sigma_{1b_j} - \Phi_{0l_i}^2 \Phi_{0ll_i} \qquad (i = 1, N_1) \qquad (15)$$

ここで A_{2ij} および A_{3ij} を決定するためには(4)式中の Φ_{1il} 項を評価する必要があり、そのために本研究では有 限差分法を用いる.以下に本手法で改良した点を述べる.



Fig. 5 Computed free-surface contours (Wigley model, Fn=0.316)

まず自由表面パネルの代表点を Fig.2 に示すような 計算座標系 $\xi - \zeta$ で定義する.計算空間 $\xi - \zeta$ では物理空

-208 -

訂 正 Fig.6とFig.8の図が入れ替わっておりました. お詫びして,訂正させていただきます.

間上の格子点が等間隔な格子点に変換されており、数 値計算を行う上で便利である.計算座標系 $\xi - \zeta$ を用い $\Phi_{1ll}を記述すると以下のようになる.$

 $\Phi_{1ll} = (\Phi_{1l_{\ell}}\xi_x + \Phi_{1l_{\ell}}\zeta_x)u_0 + (\Phi_{1l_{\ell}}\xi_y + \Phi_{1l_{\ell}}\zeta_y)v_0 \quad (16)$

ここで uo および vo は静止水面における2重模型流場の



Fig. 6 Comparison of free-surface profiles on the hull (Wigley model, Fn=0.316)

流線の方向余弦である.このとき次の関係がある.

 $\xi_x = y_{\zeta}/J \quad , \quad \zeta_x = -y_{\xi}/J$ $\xi_y = -x_{\zeta}/J \quad , \quad \zeta_y = x_{\xi}/J$ with $J = x_{\xi}y_{\zeta} - y_{\xi}x_{\zeta} \qquad (17)$

また (16) 式は (u_0, v_0) の $\xi - \zeta$ 座標における反変成分 (u^{ξ}, u^{ζ}) を用いて次のように表わすことができる.

$$\Phi_{1ll} = u^{\xi} \Phi_{1l_{\xi}} + u^{\zeta} \Phi_{1l_{\zeta}}$$
with $u^{\xi} = \xi_x u_0 + \xi_y v_0, u^{\zeta} = \zeta_x u_0 + \zeta_y v_0$ (18)

さらに上流差分を次のように定義する.

$$u^{\xi} \frac{\partial}{\partial \xi} = \frac{1}{2} (u^{\xi} + |u^{\xi}|) \delta_{\xi}^{-} + \frac{1}{2} (u^{\xi} - |u^{\xi}|) \delta_{\xi}^{+}$$
(19)

$$u^{\zeta} \frac{\partial}{\partial \zeta} = \frac{1}{2} (u^{\zeta} + |u^{\zeta}|) \delta_{\zeta}^{-} + \frac{1}{2} (u^{\zeta} - |u^{\zeta}|) \delta_{\zeta}^{+} \qquad (20)$$

このとき ($\delta_{\xi}^{-}, \delta_{\xi}^{+}, \delta_{\zeta}^{-}, \delta_{\zeta}^{+}$) は上流差分演算子である.本 研究では計算領域境界近傍の一部を除く自由表面全域 で4 点差分を採用する. Fig.2 のインデックスを用いて 差分演算子を表現すると以下のようになる.

$$\delta_{\xi}^{-}\phi_{i,j} = C_{-3}^{-}\phi_{i-3,j} + C_{-2}^{-}\phi_{i-2,j}$$

$$+ C_{-1}^{-} \phi_{i-1,j} + C_0^{-} \phi_{i,j} \tag{21}$$

$$\begin{aligned} \hat{s}_{\xi}^{+}\phi_{i,j} &= C_{+3}^{+}\phi_{i+3,j} + C_{+2}^{+}\phi_{i+2,j} \\ &+ C_{+1}^{+}\phi_{i+1,j} + C_{0}^{+}\phi_{i,j} \end{aligned}$$
(22)

$$\delta_{\zeta}^{-}\phi_{i,j} = C_{-3}^{-}\phi_{i,j-3} + C_{-2}^{-}\phi_{i,j-2} + C_{-1}^{-}\phi_{i,j-1} + C_{0}^{-}\phi_{i,j}$$
(23)

$$\delta_{\zeta}^{+}\phi_{i,j} = C_{+3}^{+}\phi_{i,j+3} + C_{+2}^{+}\phi_{i,j+2} + C_{+1}^{+}\phi_{i,j+1} + C_{0}^{+}\phi_{i,j}$$
(24)

ここで
$$\phi = (\Phi_{1l}, x, y)$$
また Cは差分係数であり、本研



Fig. 7 Computed and measured free-surface contours (Wigley model, Fn=0.267)

究では 3rd-order upwind スキームを用いた.計算領域 境界近傍では上流側に 3 点存在しない場合があるが, そのときは使用可能な全点を用いた上流差分を行って いる.また領域境界上で上流側に計算点がない場合は 下流の1点を用いて1st-order downwindスキームを採 用しており、この場合は上流差分型Radiation条件と いう目的を逸脱する.しかしその領域は全体の極一部 に限られており、後に示す計算結果を見る限り悪影響 は小さいと考えられる.以上の方法では計算空間と物 理空間が独立しており、その座標変換が一意である限 り任意のパネルトポロジーが採用できることになる.



Fig. 8 Comparison of free-surface profiles on the hull (Wigley model, Fn=0.267)

一方,自由表面パネルの構成点を求める際,著者が RaNS 方程式解法¹²⁾において開発したポアソン方程式 ソルバーを採用している.すなわち,以下の方程式を exponential スキームを用いて解く.

$$g^{11}x_{\xi\xi} + g^{22}x_{\zeta\zeta} + 2g^{12}x_{\xi\zeta} + f^1x_{\xi} + f^2x_{\zeta} = 0 \quad (25)$$

$$g^{11}y_{\xi\xi} + g^{22}y_{\zeta\zeta} + 2g^{12}y_{\xi\zeta} + f^1y_{\xi} + f^2y_{\zeta} = 0 \quad (26)$$

ここで g^{ij} は幾何係数¹³⁾, f^1 および f^2 は control function で,格子点のクラスタリングなどを制御する.本研究 では簡便のため $f^1 = 0$, また $f^2 = f^2(\zeta)$ としている が,後述するように生成された格子は船体近傍で直交 性が良く,またその領域における自由表面解像度を重 視するとすれば,全体的なパネル密度配分は妥当なも のと思われる.格子の直交性は特に格子密度の高い領 域において,座標変換に関する数値精度保持のために 重要要件になると考えられる.

3. 計算決結果および考察

既に述べたように、本研究では数値解法の評価 の第一段階として、Wigley モデルおよび Series60 $C_B=0.6$ モデルを対象船型とする. これらの船 型はITTC(International Towing Tank Conference)の CEP(Cooperative Experimental Program)¹⁴⁾で採用さ れ,過去の実験や計算例が豊富である.Blunt な船首 を持つ船型や,水面貫通翼に適用した結果は続報で報 告する予定である.以下に述べる計算結果の考察にお いて,座標および自由表面波高は船長で無次元化し, また x=(0,1) はそれぞれ (FP,AP) に対応すること,加 えて計算では姿勢変化を考慮していないことに注意さ れたい.



Fig. 9 Comparison of computed free-surface overview (Wigley model, Fn=0.267)



Fig. 10 Comparison of wavemaking resistance coefficients (Wigley model)

3.1 Wigley モデル

本研究では、まず H 型と O 型トポロジーパネル (以 下 H 型および O 型パネルと略称)の相対評価、および O 型パネルを用いた場合のパネル依存性を確認するため に、Wigley 船型を対象とした計算を行った。Wigley 船 型は次式で与えられる: $y = b[4x(1-x)][1 - (2z/d)^2]$. ここでb = B/2=0.1は船の半幅、d=0.0625は喫水であ る。Fig.3 に自由表面パネルの全景を示す。H 型パネル と O 型 Casel パネルの総数は同じ (1,800) とした. さ らに O 型 Case2 パネル (総数 2,400) および O 型 Case3 パネル (総数 1,200) も準備した. O 型 Case1~Case3 では放射方向 (ζ方向) のパネル列数が変更してある. なお船体には全ての計算で 300 パネルを用いている.



Fig. 11 Arrangement and density distribution of free surface panels (Series60 model)

H型パネルの配置方法,特に特にパネルの拡大/縮 小方法は,著者による過去の計算例¹¹⁾を参考にして決 定した.一方,O型パネルに関してはH型パネルとほ は同様な計算領域を包括できることを前提とし,前述 した f^1 および f^2 を固定して Case1~Case3 パネルを生 成した.

Fig.4 に各パネル配置におけるパネル密度(1/パネル 面積) 分布を示す. FP および AP 近傍では急激な圧力 上昇がおこるため、その領域でパネル密度を上げる必 要がある.その場合,H型パネルでは波が伝搬しない と考えられる領域(特に FP 近傍の外部領域)のパネル 密度が上昇し、全体として不効率であることが分かる. その点に関し,船体近傍領域の解像度を重視するとす れば, O型パネルを用いた方がより効率的であること が示されている.しかしながら,外部境界近傍のパネ ル密度は相対的に低くなっており、その領域では上流 差分の dissipation 影響によって波減衰が起ることが予 想できる.外部境界近傍の波減衰は数値 dissipation の 悪影響である反面,領域外端における波の反射現象を 抑制する効果も無視できない.Lungu ら¹⁵⁾は2次元 NS 方程式法による自由表面流場計算を行った際,下流 境界において強制的に鉛直方向流速成分を減衰させる 数値消波板 (numerical wave absorber) を提案し、上述 した波の反射現象を抑制することによって計算安定性 を向上させている. R-S 法の計算安定性を低下させる 原因の一つは境界端における波の反射現象であるが、 上流差分の dissipation 影響を有効に作用させることに より Lungu ら¹⁵⁾と同様な効果を利用する観点も重要

であると思われる.



Fig. 12 Computed and measured free-surface contours (Series60 model, Fn=0.316)

まず本手法の自由表面解析精度を考察する.フルード 数 $Fn = U_0/\sqrt{gL} = 0.316(L \ theorem theo$ 験値に近いが、これは船首近傍のパネル密度が相対的 に高く、自由表面解像度が向上したためであると思わ れる.また Fn=0.316 における O 型パネルの結果は船 尾近傍において H 型パネルのものより実験値に近い.



Fig. 13 Comparison of free-surface profiles on the hull (present vs. linear solutions, Series60 model, Fn=0.316)

続いて船体近傍領域における自由表面場に関して考 察する. Fn=0.316 および 0.267 における自由表面コ ンターの比較を Fig.5 および Fig.7 に示す. Fn=0.267 においては横浜国立大学で計測された実験値(YNU, Fn=0.267)¹⁴⁾も示している.船側波形では比較的目立 たなかったH型パネルとO型パネルの解像度の差が, 船体からやや離れた領域では顕著であることが分かる. 船首波系の形状に着目すると, H型パネルの結果はO 型パネルの結果よりも船体幅方向に広がった形になって おり. 0型パネルの各波の山谷が H型パネルの場合よ り明瞭である. Fn=0.267 における実験値との比較では ○型パネルの結果がより実験値の形状に近い.一方,船 首波系の頂上を連ねる線が船体中心線となす角度はH 型/O型パネル間で大差ない(共に約19°). 船首波系 の伝搬角度がほぼ同じであるに関わらず, 波の山谷が H 型パネルで緩やかになった理由として短波長成分に関 する上流差分の dissipation 影響が考えられるが, それ は dissipation は数値的な減衰効果だけでなく拡散効果 にも関連するためである. 今回の O 型パネルでは船体 近傍のパネル密度がH型より高く、その領域における dissipation 影響がより少なかったものと考えられる.

O型 Case1~Case3 の結果の差は船体近傍領域では 比較的小さいが,船体から離れた領域,特に船尾後方 において認められる.パネル密度が上がる程波の山 谷が明瞭になる傾向がある.また O型 Case1 と O型 Case2 の差は比較的小さい. Fig.9 には *Fn*=0.267 にお けるH型パネルとO型Caselパネルによって計算した自由表面形状を鳥瞰図で比較している.図中,自由表面波高はz軸方向に拡大されていることに注意されたい.鳥瞰図で見るH型パネルの波の広がりはO型Caselパネルのものに比べてより単調であり,上述した問題点を明確に示している.自由表面コンターで比較する限り,波形全体に関してO型Caselパネルの結果がより実験値に近い.一方,O型Caselパネルの結果は計算領域外端近傍でH型パネルの結果よりも減衰している様子が確認できる.これは外端近傍のパネル拡大率がH型パネルより大きく,上述した数値消波板効果がより大きいためであると考えられる.



Fig. 14 Comparison of free-surface profiles on the hull (present vs. nonlinear solutions, Series60 model, Fn=0.316)

最後に造波抵抗の推定精度について考察する. Fig.10 は今回行った全ての計算結果を実験値(IHI)¹⁴⁾と比較し たものである.実験値のCwは造波抵抗係数(k=0.031), またCwp は波形解析による造波抵抗係数であり, 計測 は sinkage および trim 共に free 状態で行われている. O型Case1~Case3の計算値の差は比較的小さく、パ ネル密度について上位2例のCase1とCase2の間で最 大約2%であった.計算値におけるハンプおよびホロー の位置はO型Case1~Case3で一致しており、特にそ のハンプの位置は実験値 Cwのものと極めて良く一致 している. またO型パネルの結果は Fn=0.316 近傍で 実験値 Cwと定量的にも一致している.一方, H型パ ネルによる結果も実験値が示す傾向を良く再現してい る. 今回のH型パネルの結果は全体的にO型パネルに よる結果より大きな値となっているが、H型パネルに よる他の計算例 (例えば田原¹⁶⁾など) ではより実験値に 近い値が予測されている場合もあることから、造波抵 抗の推定精度に関する O型/H型パネルの優劣は、現 時点では明らかではない.

一方, H型パネルを用いた際, Fn=0.350を越えた 段階で計算領域外端における波の反射現象が起り計算 が発散した.今回のO型パネルを用いた計算では高速 域(Fn=0.4 近傍)でもその波の反射現象は起きていな い.これは前述した数値消波板効果がO型パネルにお いては有効に機能したためであると考えられる.H型 パネルにおいても同様な効果を助長することは可能で あるが,その場合船体近傍のパネル密度を現状のまま とすれば,トポロジーの特性上総パネル数が増大する ことになる.



Fig. 15 Computed free-surface overview (Present O-Type, Series60 model, Fn=0.316)

3.2 Series60 C_B=0.6 モデル

本計算手法の実用船型に対する応用例として,Series60 $C_B=0.6$ モデルに関する計算結果を以下で考察する. Fig.11に自由表面パネルの全景,およびパネル密度分 布を示す.使用パネルはO型とし,パネル数はWigley モデルに関して行った計算結果をもとにO型 Case1 と 同等にしたが,それは造波抵抗および船体近傍領域の 計算精度に関し,パネル密度上位2例のO型 Case1 と Case2 で大きな差が見られないためである.自由 表面場の解析精度を評価するために,Fn=0.316に おける計測値^{17),18)},さらに他の計算手法による結果 として SPLASH(Rosen³⁾,自由表面パネル数1000), Raven¹⁹⁾(自由表面パネル数約 3000)の線形および非線 形解,そして RaNS 方程式法による著者らの解^{2),20)}も 用いる.それら計算値のパネル/格子依存性に関する 評価は文献^{19),20),21)}で行われている.ここで Rosen³⁾ の方法では O 型パネルを, また Raven¹⁹⁾の方法では H 型パネルを共に有限差分型 Radiation 条件で用いてい る. Fig.12~14 には自由表面コンターおよび船側波形 の比較を示す.

まず船側波形について考察する。Fig.13 では本計算 による解 (Present, O-Type) を他の線形解 (SPLASH, Raven-linear) および実験値 (IIHR) と比較している. ここで示す計算値は全て線形自由表面境界条件の解で あるが、振幅および位相に関して実験値と比較的良く 一致している. Fig.13 に示す全ての計算で船首波のハ ンプおよびホローの振幅をやや過小評価する傾向があ り、これは先に Wigley モデルの計算結果でも指摘し た非線形影響の欠如によるものと思われる。船首波の ハンプ近傍および肩波のホローについては本計算結果 が他の結果より若干実験値に近いが、船側波形に関し ては全体的に同等な予測精度を有していると思われる. Fig.14 では本計算による解を Raven¹⁹⁾の非粘性流法に よる非線形解 (Raven-nonlinear), 著者らによる RaNS 方程式法による非線形解 (Tahara et al.)^{2),20)}, および 実験値(IIHR)と比較している.非線形解では上述の 船首波のハンプおよびホローの振幅が本計算結果より も大きく,そしてより実験値に近い.この部分の予測 精度を高めるためには非線形影響の考慮が必要である と考えられる.



Fig. 16 Comparison of wavemaking resistance coefficients (Series60 model)

次に船体近傍の自由表面解析精度 (Fig.12) に関して 考察する.本計算結果は実験値と比較的良く一致して いる.特に肩波系の生成,およびその船首波系と干渉 する様子 (x=0.7 近傍) は SPLASH の解では明確でな いが,本計算結果では詳細に予測されている.船首波 系の頂上を連ねる線が船体中心線となす角度は本計算 結果および Raven の線形解でほぼ同じであり (約19°), 実験値のものと比べるとやや過大評価されている.一 方,その角度はRaNS 方程式法およびRavenの非線形 解においてより実験値に近い.Ravenの解法は自由表 面条件以外は同一であり,さらに線形解を非線形解が 改良していることから,本計算手法に非線形影響を取 り入れ,それによって解が改善できるかどうかを今後 検討する必要がある.Fig.15 には本計算結果の自由表 面鳥瞰図を示す.本論文で示す他の計算結果の同様な 鳥瞰図については著者の文献²⁾を参照されたい.今回 の計算値は後方外部領域においてWigleyの場合と同 様に波高減衰が認められるものの,多くの観点で実験 値に類似している.

最後に Fig.16 において本計算で得られた造波抵抗係数を 実験結果¹⁴⁾と比較する.実験は IHI および日本鋼管 (NKK) で行われ,図中の Cr-ITTC は ITTC correlation line を 用いて算出した剰涂抵抗係数,また Cw-Sch. は Schoenherr line と Form factor(k=0.090:IHI, =0.130:NKK) を用い て算出された造波抵抗係数であり,計測は全て sinkage および trim 共に free 状態で行われている.本計算結果 の定性的傾向は実験値と良く一致している. Fn=0.33 近傍のハンプではやや過小評価する傾向が確認できるが (Cw-Sch.-IHI に対し約 15%),Fn=0.31 より低い Fn領域では Cw-Sch.-IHI と定量的にも良く一致している. 一方,Fn=0.35 以上では全体的に過小評価する傾向も 指摘できる.今回の計算は船体の姿勢変化を考慮せず 全て fix 状態で行っているが,定量的精度向上のために はその影響も考慮する必要があると考えられる²¹).

4. 結言

本報告では Dawson-type R-S 法を任意の自由表面 パネルトポロジーに対応できるように改良した.自由 表面条件は荻原ら¹⁰⁾が提案した非線形条件式に基づい ているが、本報告ではその第一近似解、すなわち従来 多く用いられている Dawson が提案した線形近似条件 の解を示した.自由表面パネルには従来の H 型に加え O型トポロジーも採用し、O型トポロジーパネルの生 成にはポアソン方程式ソルバーを用いた.本報告で は、手法評価の第一段階として Wigley モデルおよび Series60 $C_B=0.6$ モデルを対象船型とした.本報告の 結論をまとめると以下のとおりである.

1. 物理空間で定義された自由表面条件式の一部を計 算空間に変換し、その変換座標系で上流差分法を 定義することにより、自由表面パネルトポロジー に制限されない R-S 法の適用が可能となった.こ の方法は現存する各 R-S 法において比較的簡単な 計算コード変更で導入可能なものである.

- 今回採用したO型トポロジーパネルは、従来多く 用いられているH型トポロジーパネルに比べ、船 体近傍のパネル密度を重視するという観点では極 めて有効であることが確認された.またBluntな 船首にも対応しやすいと考えられるが、その評価 は今後の課題として残された.
- 本手法をWigley モデルに対して応用した結果, 船体からやや離れた領域(約半船長以内)の自由表 面解析精度は,船首波系/肩波系の生成・干渉の 予測に関し,O型トポロジーパネルの使用によっ て大きく改善されることが分かった.これはパネ ル密度の改善による dissipation 影響の抑制に関 連すると考えられる.
- 本手法をO型トポロジーパネルを用いてSeries60 C_B=0.6 モデルに対して応用した結果,本手法は 船側波形および船体近傍領域の自由表面場に関し て満足できる精度を有することが確認された.し かしながら,線形近似条件を用いていることの限 界と考えられる点も指摘され,非線形影響の考慮 は今後の課題として残された.
- 本手法によって予測した造波抵抗は、Wigley モデルおよび Series60 C_B=0.6 モデルに関し、定性的に実験値と一致した.また Fn 領域によっては定量的にも実験値と良く一致した.本研究で計算を行った限りでは O型トポロジーパネルの精度がH型トポロジーパネルを上回ったが、それに関する詳細な評価は今後さらに行われるべきである. 一方、今回用いた O型トポロジーパネルでは高速域(Fn=0.4 近傍)でも安定した計算が可能であったが、これは領域外端近傍で拡大されたパネル配置による数値消波板効果によるものと考えられる.

O型トポロジーパネルを用いた計算結果は、Wigley モデルおよび Series60 $C_B=0.6$ モデルに関して妥当な 結果を与えることが示されたが、本計算法の最も重要 な利点は任意のパネルトポロジーが物体形状に応じて 選択できることである.上述した Blunt な船首を持つ 船型に対する応用などを通して、本計算手法をより詳 細に評価する必要がある.自由表面条件式の非線形影 響や船体姿勢変化の考慮も含め、今後の研究でさらに 検討しく予定である.

最後に,本研究を行うにあたり,種々のご助言ご協 力をいただきました大阪府立大学工学部姫野洋司教授, 並びに九州大学工学部安東潤助教授に感謝の意を表わ します.

参考文献

1) Dawson, C.W.: A Practical Computer Method

for Solving Wave Problems, Proc. 2nd International Conference on Numerical Ship Hydrodynamics, Berkeley, 1977, pp. 30-38

- Tahara Y.: Evaluation of a RaNS Equation Method for Calculating Ship Boundary Layers and Wakes Including Wave Effects, 日本造船学会 論文集, 第180 号, 1996, pp. 59-80
- Rosen, B.: SPLASH Free-Surface Code: Theoretical/Numerical Formulation, South Bay Simulations Inc., Babylon, NY, private communication, 1989
- 江口辰哉: Rankine source 法による定常造波問題の数値解法-特に collocation method の適用について-,日本造船学会論文集,第177号,1995, pp. 101-112
- 5) 中武一明,安東潤,市川卓示,片岡克己:幅広船ま わりの波流れの一計算法,西部造船会々報,第90 号,1995, pp. 21-29
- (6) 安東潤, 中武一明: Rankine Source 法による波流 れの一計算法, 西部造船会々報, 第75号, 1987, pp. 1-12
- Jensen, P.S.: On the Numerical Radiation Condition in Steady-State Ship Wave Problem, J. Ship Research, Vol. 31, No. 1, 1987, pp. 14-22
- 8) Jensen, G., Soding, H., and Mi, Z.-X.: Rankine Source Methods for Numerical Solutions of the Steady Wave Resistance Problem, Proc. Sixteenth Symposium on Naval Hydrodynamics, Berkeley, 1986, pp. 575-582
- 9) 瀬戸秀幸: 定常造波問題における Rankine Source 法の基礎と開境界処理に関する一考察,西部造船 会々報,第81号,1991, pp. 11-28
- 萩原誠功, 丸尾孟: 船体まわりの自由表面流れの非 線形計算法, 日本造船学会論文集, 第157号, 1985, pp. 34-46
- 11) Tahara, Y. and Longo, J.: Nonlinear Free-Surface Flow around a Yawed Ship, Boundary Element Technology IX, Computational Mechanics Publications, 1994, pp. 121-128
- 12) Tahara, Y.: Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments, 関西造船協会誌, 第 220 号, 1993, pp. 29-47
- 13) Thompson, J.F., Warsi, Z.U.A., and Mastin, C.W.: Numerical Grid Generation, Elsevier Science Publishing Co., Inc., 1985

- 14) ITTC: Report of the Resistance and Flow Committee, Proc. 18th Int. Towing Tank Conference, Kobe, Japan, 1987
- 15) Lungu, A. and Mori, K.: A Study on Numerical Schemes for More Accurate and Efficient Computations of Free-Surface Flows by Finite Difference Method, 日本造船学会論文集, 第173 号, 1993, pp. 9-17
- 16) Tahara, Y.: A Boundary-Element Method for Calculating Free-Surface Flows around a Yawed Ship, 関西造船協会誌, 第 218 号, 1992, pp. 55-67
- 17) Toda, Y., Stern, F. and Longo, J.: Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60C_B=.6 Ship Model Part 1: Froude Numbers .16 and .316, J. of Ship Research, Vol. 37, No. 4, December 1992, pp. 360-377
- 18) Longo, J., Stern, F. and Toda, Y.: Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series $60C_B=.6$ Ship Model -Part 2: Scale Effects on Near-Field Wave Patterns and Comparisons with Inviscid Theory, J. of Ship Research, Vol. 37, No. 1, March 1993, pp. 16-24
- 19) Raven, H.C.: Nonlinear Ship Wave Calculations Using the RAPID Method, Proc. 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, Iowa, 1993, pp. 95-118
- 20) Tahara Y. and Stern, F.: A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number, Proc. CFD Workshop Tokyo 1994, Tokyo, Vol. 1, March 1994, pp. 45-55; also J. Computational Physics, Vol. 127, 1996, pp. 398-411
- 21) Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994

論

[討論] (日立造船技術研究所)田中 寿夫

討

従来のH型格子を用いたRankine source 法では, Froude 数が低い場合には素成波の波長が短くなるため,パネルのサイズを小さくせざるを得ず,結果的に 多大のパネル数を要することとなり,実用的とは言い 難い面がありました.一方,本論文で示された計算法 では,船体近傍に効率的にパネルを集中させることが できるので,上述の問題点をある程度解決できると思 えますが,著者の見解をお聞かせください.

[回答]

ご指摘のとおり、本研究で用いたO型パネルは船体

近傍の自由表面解像度を重視するという観点では、H 型パネルより有効ではないかと考えています. Series60 $C_B=0.6$ 船型に対する適用では、Fn=0.2近傍の低速 域で造波抵抗の計測値が示すハンプ/ホローを定性的 に捉えることができています.本研究で示した改良方 法は現存する各 Rankine source 法に比較的簡単に導入 できるものと思われ、設計ツールの機能向上に少しで もお役に立てば幸いに存じます.ご討論ありがとうご ざいました.

[討論] (大阪府立大学)西尾 茂

造波抵抗の推定精度についての考察の中で,H型パ ネルの場合の計算領域外端における反射波の影響とO 型パネルの場合の波の減衰が同等に精度評価の要因と して扱われていますが,推定精度の評価という観点か らはこれら2つの要因を対等に扱うことは難しいよう に思われますがいかがでしょうか.

[回答]

ご討論ありがとうございます.等方性特異点分布を 有限領域で用いる Rankine-source 法では,領域外端位 置およびフルード数条件によって領域外端における波 反射が起こることがあり,手法の安定性を低下させる ことが知られています.本研究では、この現象を回避 する一つの方法として, Lungu ら¹⁵⁾が提案した数値消 波板 (numerical wave absorber)の概念は有効なのでは ないかと考えました.本研究で用いた〇型パネルは H 型パネルよりも船体近傍で要素密度が高く、逆に領域 外端近傍ではそれが低くなっています. 従って船体近 傍の解像度が高い反面,域外端近傍における O 型パネ ルの計算精度は相対的に低く (dissipation がより大き く) なっていることにより、上述した数値消波板に類似 する効果がより強くなったと考えられます. これがО 型パネルの計算安定性につながっていることは事実と 思われるのですが、同時に外部領域における計算精度 低下というマイナス点も十分考慮して応用するべきだ と思います.

-216-

非H型トポロジー自由表面パネルを用いた Dawson-Type Rankine-Source 法による定常造波問題の数値計算 一第二報:Blunt nose への適用性 および非線形自由表面条件の応用についてー*1

正会員田原裕介*2

A Numerical Approach for Steady Ship-Wave Problem Based on Dawson-Type Rankine-Source Method with Non-H-Type-Topology Free-Surface Panels

- 2nd Report: application to blunt-nose body and nonlinear free-surface boundary conditions -

By Yusuke TAHARA (Member)

This paper presents a numerical approach for steady ship-wave problem, which is based on extension and modification of Dawson-type Rankine-source method in order to utilize arbitrary topology of free-surface panels. In this 2nd report, the main emphasis is placed on application of the present method to blunt-nose body and consideration of nonlinear effects into the free-surface boundary conditions. The computational results are presented and discussed for surface piercing NACA0024 foil, and Wigley and Series60 $C_B=0.6$ ship models including detailed comparisons with experimental data. It appears that the present method adequately predicts free-surface flows around blunt nose, and introduction of nonlinear free-surface effects clearly improves linear solutions especially for forebody wave fields.

Keywords : Steady Ship-Wave Problem, Rankine-Source Method, Arbitrary Topology Panels, Nonlinear Free-Surface Boundary Conditions, Wigley Model, Series60 Ship Model, Surface Piercing NACA0024 foil

1. 緒言

定常造波問題の解法としての Rankine-source 法(以下 R-S 法と略称)は計算労力が RaNS(Reynolds-averaged Navier-Stokes) 方程式法に代表される粘性流法より少 なく,妥当な精度で自由表面流場を計算できることが 大きな利点である^{1),2)}. Dawson³⁾以来, R-S 法に関す る多くの研究が為されてきたが,汎用性のある船型デ ザインツールとなるためには,幾つかの解決すべき課 題が残されている¹⁾. その一つは,自由表面パネルト ポロジーに関する制限であった.本研究ではこの問題 点に着目し,自由表面パネルトポロジーに制限されな い R-S 法の開発を試み,従来用いられてきた H 型以外 のトポロジーを導入することにより,特に船体近傍自 由表面場の解像度向上を目指している.前報¹⁾では本 手法の概要を述べ,2種の供試船型に対する結果を検 討し,従来の R-S 法では適用できなかった O 型トポロ ジーパネルの有効性を示した.本報では Blunt nose 物 体への適用性,および非線形自由表面影響の考慮によ る解の改善について検討する.

^{*1} 平成10年5月20日 関西造船協会春季講演会におい て講演,原稿受付 平成10年6月10日

^{*2} 大阪府立大学工学部



Fig. 1 Definition sketch of coordinate system

2. 基礎方程式および数値計算方法

本手法の基本部分は前報¹⁾と同一であるので、ここでは非線形自由表面条件の適用に焦点を絞り、数値計算法の概略を述べる.まず前報¹⁾と同様、デカルト座標系をFig.1のように定義し、一様流 $\vec{U}_{\infty} = (U_0, 0, 0)$ 中に固定された船体を考える.自由表面境界条件には、 荻原ら⁴⁾が導いた以下の式を用いる.

$$\Phi_{0l}^2 \Phi_{1ll} + 2\Phi_{0l} \Phi_{0ll} \Phi_{1l} + g \Phi_{1z}$$

= $-\Phi_{0l}^2 \Phi_{0ll} + g D_1 - \Phi_{0l} D_{2l}$ (z = 0) (1)

$$D_1(x,y) = \Phi_{1x}(\eta_x - \eta_{0x}) + \Phi_{1y}(\eta_y - \eta_{0y})$$

$$-(\Phi_{0zz} + \Phi_{1zz})\eta \tag{2}$$

$$D_2(x,y) = -\frac{1}{2g} (\Phi_{1x}^2 + \Phi_{1y}^2 + \Phi_{1z}^2)$$
$$-\frac{1}{g} [(\Phi_{0x} + \Phi_{1x})\Phi_{1xz} + (\Phi_{0y} + \Phi_{1y})\Phi_{1yz}$$

$$+\Phi_{1z}\Phi_{1zz}]\eta\tag{3}$$

$$\eta_0(x,y) = \frac{1}{2g} (U_0^2 - \Phi_{0x}^2 - \Phi_{0y}^2) \tag{4}$$

ここで $\eta(x,y)$ は自由表面波高, Φ_0 は2重模型速度ポテ ンシャル, Φ_1 は自由表面流れを表わす速度ポテンシャ ル,lは2重模型流れの流線座標, D_1 および D_2 は η の 高次項を省略した Φ_1 に関する非線形項である.前報¹⁾ では D_1 および D_2 をゼロとした第一近似式,すなわち Dawson³⁾が導いたものと同一の線形自由表面条件式 を用いた.本研究では非線形影響項 $D_1 \cdot D_2$ を考慮する ために, 萩原ら⁴⁾も用いた反復計算法を導入する.そ こで前報¹⁾のマトリクス方程式(14)(15)を以下のよう に定義する.

$$2\pi\sigma_{1b_i}^{(n)} + \sum_{j=1}^{N_0} A_{0_{ij}}\sigma_{1b_j}^{(n)} = \sum_{j=1}^{N_1} A_{1_{ij}}\sigma_{1f_j}^{(n)}$$
(5)



Fig. 2 Panel arrangement on free surface (surface piercing NACA0024 foil)





$$\sum_{j=1}^{N_1} A_{2_{ij}} \sigma_{1f_j}^{(n)} - 2\pi g \sigma_{1f_i}^{(n)}$$
$$= \sum_{j=1}^{N_0} A_{3_{ij}} \sigma_{1b_j}^{(n)} - \Phi_{0l_i}^2 \Phi_{0ll_i} + g D_{1_i}^{(n-1)} - \Phi_{0l_i} D_{2l_i}^{(n-1)}$$
(6)

上式中の (*n*) は Iteration index である.その他の変数 定義および index については前報¹⁾を参照されたい.

ここで $D_1 \cdot D_2 \varepsilon$ ゼロとした場合の解を第一近似解 (n = 1), すなわち線形解とする.第二近似解以降を求 めるためには, (6) 式の $D_1^{(n-1)}$ および $D_2^{(n-1)}$ を前段階 (n - 1)の解を用いて計算することになる.このとき (2)-(4) 式中の微分項を評価する必要があり,本研究で はそれらを全て中心差分で求めている.その差分は 前報¹⁾と同じく計算座標系で行うため,自由表面パネ ルのトポロジーには依存しない.さらに本研究では (2)-(4) 式中の各微分項のオーダーを調べ, $D_1 \cdot D_2$ の 簡略化の可能性も検討した.その結果は後述する.



Fig. 4 Comparison of wave profiles on body surface (surface piercing NACA0024 foil, Fn=0.340)



Fig. 5 Comparison of free-surface contours between present linear and nonlinear solutions (Wigley model, Fn=0.316, nonlinear:n=2)

荻原ら⁴⁾ は, (1)-(4) 式を用いた反復計算の結果は, 必ずしも収束解に至らない可能性を指摘している. さ らに荻原ら⁴⁾は領域緩和係数を導入し,計算を安定化す ることも検討した.本研究では,領域緩和係数を用いな くても,第二近似非線形自由表面条件 (n = 2)によって 線形解が十分改善されることがわかり,よってその後の 繰返し計算は行っていない.以下で示す全ての非線形 解は,第二近似非線形自由表面条件 (n = 2)のもので ある.より高次な近似条件の検討は今後の課題とする.

3. 計算決結果および考察

3.1 Blunt Nose 物体への適用性

本研究では、まず NACA0024 水面貫通翼を供試模型とし、従来 H型自由表面パネルでは困難であった



Fig. 7 Comparison of wavemaking resistance coefficients (Wigley model, nonlinear:n=2)

Blunt nose 形状物体への適用性を評価した.自由表面 条件には前報¹⁾と同じ線形近似条件を用いた.自由表 面パネルの全景および翼前縁近傍を Fig.2 に示す.パ ネルトポロジーは O 型である.前報¹⁾で行った,解の パネル密度依存性に関する検討の結果に従い,自由表 面パネル数は 1,800 とした.一方,翼の水深方向スパ ンは 1 翼弦長とし,その表面には 300 パネルを配置し た.またスパン長をそれより増加しても,解への影響 は無視できることを確認した.

本供試模型に従来の方法で H 型パネルを適用すれ ば,流線方向のパネル配列が前縁近傍でねじれ,計算 精度および安定性を低下させる.その類似した具体 例は,荻原ら⁴⁾の結果 (Case B 船型)にも示されてい る.本計算で用いた O 型パネルは無理なく物体に適 合し,前縁近傍の格子直交性も良い.またフルード

-219-



Fig. 8 Comparison of free-surface contours between present linear and nonlinear solutions (Series60 model, Fn=0.316, linear:n=1, nonlinear:n=2)

数 $Fn = U_0 / \sqrt{gL} = 0.150 \sim 0.550 (L は 翼弦長) で計算を$ 行った結果、外境界波反射などによる解の発散が生じ ないことを確認した. Fig.3は Fn=0.370 における前 縁近傍の自由表面コンターの計算結果である.座標お よび自由表面波高は翼弦長で無次元化し, X=(0,1)は 翼の (前縁:LE,後縁:TE) に対応することに注意された い.一般に Blunt 船首近傍で観測される,急上昇する 自由表面や,円弧状に広がる自由表面コンターが再現 されている. Fig.4 には翼側面における自由表面波高 の実験値⁵⁾(Rn=1,520,000)との比較を示す.実験では X=0.4 近傍より後方で境界層剥離が観測されている. X=0~X=0.4の領域では、計算値と実験値は良く-致しているが, X=0.4 より後方では明らかな差がみら れる.これは本計算手法には粘性影響や剥離影響が 考慮されていないためである.しかしながら,Blunt nose 近傍の自由表面場の予測という観点では、本計算 法は妥当な精度を有していることがわかる.

3.2 非線形自由表面影響

ここでは非線形自由表面条件を適用し、線形解の改善を検討した結果について述べる.供試模型は前報¹⁾でも用いたWigleyモデルおよびSeries60 C_B =0.6モデルである.自由表面パネルはO型とし、その総数は前報¹⁾の結果に従い1,800、船体表面上のパネル数は300とした.前報¹⁾と同様、本研究においても模型の姿勢変化は考慮していない.また以下で用いるWigleyモデルおよびSeries60 C_B =0.6モデルに関する実験値は、前報¹⁾で用いたものと同一であり、実験状態その他詳細については前報¹⁾を参照されたい.加えて、座標および自由表面波高は船長で無次元化し、X=(0,1)



Fig. 9 Comparison of wave profiles on the hull (Series60 model, Fn = 0.316, nonlinear:n = 2)



Fig. 10 Comparison of $D_1^{(1)}$ and $D_2^{(1)}$ contours (Wigley model, Fn=0.316)

は船体の(FP,AP)に対応することに注意されたい.

まず Wigley モデルについて, Fn=0.316 において 計算した線形/非線形解の比較を Fig.5 に示す.線形解 (n=1) と比較して,非線形解の波高,特に船首波系の 波頂はより高くなる傾向がある.船側波形による比較 (Fig.6) では,特に船首近傍において,非線形解が線形 解を改善していることが示されている.船側波形にお ける線形/非線形解の違いは,X = 0.6 近傍以前で顕著 である.その後方で線形/非線形解の差が小さいのは, D_1 および D_2 項のオーダーが,主に波振幅のオーダー に依存しており,波高が比較的小さい領域では非線形 影響が小さいためである.Fig.7 は造波抵抗の実験値と の比較を示す.全体的に非線形解は線形解より低い値 を示している.ハンプおよびホローの位置は線形/非線 形解の間でほぼ同じであり,ともに実験値のものと一 致している.



次に Series60 C_B=0.6 モデルに関する計算結果を 示し,本手法を実用船型に適用した結果を考察する. Fig.8 に Fn=0.316 における線形/非線形解の比較を示 す. Wigley モデルの場合と同様,特に船首波系の波頂 は非線形解の方がより高くなる傾向がある.船首波系 の頂上を連ねる線が船体中心線となす角度は、線形/非 線形解の間でほぼ同じであり (約 19°), 実験値のもの よりやや大きい、この点に関しては、非線形解は線形 解を改善するには至っていない.一方, Fig.9 に示す船 側波形では,特に船体前半部において,非線形解が 線形解を改善していることが示されている. X =0.25 ~0.4 の範囲では Raven⁶⁾の非線形解と類似している が, それより前方では差が見られ, 船首波波頂までは Raven⁶⁾の非線形解が, さらにその前方から FP までは 本計算の非線形解がより実験値に近い、以上の結果よ り、今回用いた第二近似非線形自由表面条件は、特に 船体前半部の船側波形に関し、明らかに線形解を改善 することがわかった.

3.3 非線形項の評価

最後に、非線形自由表面条件式の各項のオーダーを 調べ、より簡便な表現式を検討した結果について述べ る. Fig,10 は Fn=0.316 における、Wigley モデル近傍 の $D_1^{(1)}$ および $D_2^{(1)}$ のコンターである。対応する自由表



Fig. 13 Comparison of free-surface profiles on the hull between original and modified nonlinear free-surface boundary conditions (Wigley model, Fn=0.316, nonlinear:n=2)

面計算結果は Fig.5 に示してある. $D_1^{(1)}$ および $D_2^{(1)}$ の影 響が顕著である領域は,船首波系および船尾波系がつら なる領域とほぼ一致している.一方, $D_1^{(1)}$ および $D_2^{(1)}$ の船側分布を,線形/非線形解の差が明らかであった 船体前半部に着目し,Fig.11 および Fig.12 に示す.こ こで $T1 \sim T7$ を以下のように定義する: $T1 = \Phi_{1x}(\eta_x - \eta_{0x})$: $T2 = \Phi_{1y}(\eta_y - \eta_{0y})$: $T3 = (\Phi_{0zz} + \Phi_{1zz})\eta$: $T4 = (\Phi_{1x}^2 + \Phi_{1y}^2) + \Phi_{1z}^2)/2g$: $T5 = (\Phi_{0x} + \Phi_{1x})\Phi_{1xz}\eta/g$: $T6 = (\Phi_{0y} + \Phi_{1y})\Phi_{1yz}\eta/g$: $T7 = \Phi_{1z}\Phi_{1zz}\eta/g$. Fig.11 および Fig.12 より,非線形項として支配的なものは T1,T3, さらに T5であることがわかる.この結果に従い, T1,T3,T5を用いた D_1 および D_2 の簡略化表現, D_1^* お よび D_2^* を以下のように表す.

$$D_1^*(x,y) = \Phi_{1x}(\eta_x - \eta_{0x}) - (\Phi_{0zz} + \Phi_{1zz})\eta \qquad (7)$$

$$D_2^*(x,y) = -\frac{1}{g}(\Phi_{0x} + \Phi_{1x})\Phi_{1xz}\eta$$
(8)

上式による非線形解と,(2)(3)式による非線形解の比較を Fig.13 に示す.両者の解はほぼ一致し,ここで求めた D₁および D₂の簡略化表現によっても,線形解を改善する非線形解を得ることがわかった.

4. 結言

本報告では,前報¹⁾において課題とされた Blunt nose を有する物体への適用,および非線形自由表面影響の 考慮を試みた.前者を検討するために NACA0024 水 面貫通翼に対して計算を行い,さらに実験値と比較し た結果,本手法は Blunt な翼前縁近傍でも安定した計 算が可能であり,妥当な計算精度を有していることを 確認した.理論上の限界により砕波現象には対応でき ないが,肥大船などのBluntな船首近傍で見られる, 急激に上昇する自由表面を,妥当な精度で推定するこ とも可能であると考えられる.一方,後者を検討す るために,前報¹⁾でも使用したWigleyモデルおよび Series60 C_B =0.6 モデルに対して非線形解を求めた結 果,第二近似非線形自由表面条件を用いることによっ て,船側波形の線形解が明らかに改善されることを確 認した.さらに非線形影響項中の各項のオーダーを調 べ,より簡便な式を求め,その式による非線形解は元 の非線形解とほぼ一致することを確認した.

最後に,本研究を行うにあたり,種々のご助言ご協 力をいただきました大阪府立大学工学部姫野洋司教授 に感謝の意を表わします.

参考文献

- 田原裕介:非H型トポロジー自由表面パネルを用 いた Dawson-Type Rankine-Source 法による定常 造波問題の数値計算-第一報:特にO型自由表面 パネルの適用について-,関西造船協会誌,第228 号,1997, pp. 79-90
- Tahara Y.: Evaluation of a RaNS Equation Method for Calculating Ship Boundary Layers and Wakes Including Wave Effects, 日本造船学会 論文集, 第 180 号, 1996, pp. 59-80
- Dawson, C.W.: A Practical Computer Method for Solving Wave Problems, Proc. 2nd International Conference on Numerical Ship Hydrodynamics, Berkeley, 1977, pp. 30-38
- 4) 荻原誠功, 丸尾孟: 船体まわりの自由表面流れの非 線形計算法, 日本造船学会論文集, 第157号, 1985, pp. 34-46
- Zhang, Z.J. and Stern F.: Free-Surface Wave-Induced Separation, J. Fluids Engineering, Vol. 118, 1996, pp. 546-554.
- 6) Raven, H.C.: Nonlinear Ship Wave Calculations Using the RAPID Method, Proc. 6th International Conference on Numerical Ship Hydrodynamics, Iowa City, Iowa, 1993, pp. 95-118

討 論

[討論] (横浜国立大学工学部)鈴木 和夫

1. 萩原らの論文では自由表面条件の非線形影響を 考慮して造波抵抗を計算する際に,圧力積分の修正項 として船側波形による浸水面の変化に対応する水線ま わりの線積分を加えています.このような考慮の必要 はないのでしょうか.

2. 貴論文の第一報を参考に我々も同様なコードを 作成し,実用船型への適用について検討しています. 以前に負の造波抵抗値を示した船型も概ねそのよう なことがなくなり実用性は高くなったのですが,肩の 張っているような肥型船の場合には低速でまだ負の造 波抵抗を示すことがあります.これらの船型の場合に は肩の部分のパネル配置を密にして低速で圧力積分が 0となるようにする等のノウハウが必要になりますの で,今回の手法についてもそのような検討が必要と思 われますがいかがでしょうか.

[回答]

1.本研究では、ご指摘の線積分を考慮していません。本研究で定義する非線型影響は、自由表面境界条件の非線型修正項の影響のみと限定したためです。

2.残念ながら、本手法をご指摘のような船型に応 用したことがなく、ご質問に対する的確な回答を行う 十分な情報を持っておりません.一方、私見としまし ては、本手法の有効性は、特に船体近傍の自由表面場 解析精度の向上にあり、ご指摘の負の造波抵抗値問題 を解決するには、別の観点にたった改良策が必要であ ろうと思います.ご指摘にあった船体パネル配置に関 する工夫も有効と考えられますし、また高次パネルの 適用も検討されるべきではないでしょうか.

貴重なご討論ありがとうございました.

関西造船協会誌 第229号 平成10年3月

CFD による二次元翼形状改良問題に関する研究^{*1}

正会員田原裕介*2,正会員姫野洋司*2

A Study on Form Optimization Problem Based on CFD for Two-Dimensional Wing Section

By Yusuke TAHARA (Member) and Yoji HIMENO (Member)

This paper presents a study on form optimization problem of two-dimensional wing section, with main focus placed on improvement of lift-to-drag ratio at a given angle of attack. The fully-elliptic Reynolds-averaged Navier-Stokes and continuity equations are solved with zero-equation turbulence model in order to compute lift and drag forces acting on the body, using a regular grid, finiteanalytic discretization, and a PISO-type velocity-pressure coupling algorithm. An overview is given of the present numerical approach, and results are presented for modification of NACA0012 wing section, including discussions regarding influence of type of form modification function and number of design parameters on optimized solutions. In addition, relative computational performance is compared between two nonlinear programming algorithms, i.e., successive linear programming (SLP) and successive quadratic programming (SQP).

Keywords: Two-Dimensional Wing Section, Form Optimization, Nonlinear Programming, RaNS Equations

1. 緒言

近年のCFD (Computational Fluid Dynamics) 技 術の進歩により、2次元および3次元物体周りの粘性 流場予測精度は実用レベルにまで高められつつある ²⁾⁻³⁾.次の課題はCFD手法の物体形状改良/最適 化への応用であろう. そのような試みは近年盛んに なりつつあり、それらは鈴木ら4)によって詳しく解説 されている.本研究では, RaNS (Reynolds-averaged Navier Stokes) 方程式の数値解に基づく3次元船体の 形状最適化問題への適用の第一段階として,著者の RaNS コードを用いて二次元翼の揚抗比改良問題を解 いた.有限解析法および PISO 型速度場・圧力場結合 法による粘性流場解法を最適化手法と結合し、一定迎 角をもって流場に置かれた基本翼の揚抗比を改良する 翼形状を求める.形状変更関数および形状変更パラ メータ数が解に与える影響を検討するとともに,幾つ かの非線形最適化アルゴリズムを用いた場合の計算効 率および収束性についても議論する.

2. 数値計算方法の概略

対象としたのは2次元翼 (NACA0012) であり,与え られた迎角およびレイノルズ数において揚抗比を改善 する翼形状を求める.まず座標系をFig.1 に示すよう に定義し,一様流 $\vec{U}_0 = (U_0, 0)$ 中に迎角 α で固定された 場合を考える.さらに形状変更関数を定義する.形状 変更関数は任意に設定できるが,今回は例として次の 2つの関数を考える.

$$B(S) = (1 - \beta_1 \cos(\pi S/S_0)(1 - \beta_2 \cos(2\pi S/S_0)))$$

$$\dots (1 - \beta_n \cos(n\pi S/S_0)) \tag{1}$$

$$B(S) = \beta_1 f_1(S) + \beta_2 f_2(S) + \dots + \beta_n f_n(S)$$
(2)

ここで S_0 は翼の後縁から時計周りに一周したときの Sの値,また $f_1(S), ... f_n(S)$ は補間関数であり区分的 線形関数やスプライン関数等を用いて表現できる. $\beta_1, ... \beta_n$ は形状変更パラメータである.

粘性流場の基礎方程式は2次元非定常/非圧縮流体 に関する RaNS(Reynolds-averaged Navier Stokes) 方 程式であり, Baldwin-Lomax 乱流モデルを用いてレイ ノルズ応力項を与える.このとき流場全域で乱流場を

^{*1} 平成9年11月15日造船三学会秋季連合大会において 講演,原稿受付 平成9年12月1日

^{*2} 大阪府立大学工学部

Computation	No. of Des.	Constraint	Iteration	Reynolds No.	Incidence	Computation	al efficiency
Case No.	parameters	Area Const.	Number	Rn	angle(deg.)	Hrs.	Ratio
Orginal				2x10 ⁶	5.	0.32	1
Case 1	3	No	30	2x10 ⁶	5.	36.48	112.59
Case 2	10	No	30	2x10 ⁶	5.	102.00	314.81
Case 3	20	No	28	2x10 ⁶	5.	210.96	651.11
Case 4	10	Yes	30	2x10 ⁶	5.	100.06	308.83
Case 5	10	Yes	100	2x10 ⁶	5.	340.80	1051.85

Table 1 Computational condition



Fig. 1 Definition sketch of coordinate system



Fig. 2 Comparison of computed and measured lift-to-drag ratios (NACA0012, Rn=3,000,000)

仮定する.さらに支配方程式を数値的に生成した非直 交曲線座標系(O型トポロジー格子)で表現し,輸送方 程式の離散化には有限解析法を,さらに速度場-圧力 場結合には PISO タイプ/1 ステップ法を採用する⁵⁾.

Fig.2 は計算格子数と揚抗比の関係を示す.対象としたのはNACA0012 翼である.ここで示す計測値を求めた実験⁶⁾では翼前縁において乱流促進がなされておらず,抵抗値も Schoenherr の平板値より低い.一方,計算では流場全域を乱流場と仮定しており,全体的に計算結果の抵抗値が実験値のものより大きく,そのため



Fig. 3 Comparison of computed and measured drag coefficients ($Rn=6,000,000, \alpha=0^{\circ}$

揚抗比の計算値は実験値より低くなっている.格子密 度が低くなるほど揚抗比は低く計算される傾向が見ら れる.しかしながら,ここで示した全ての計算結果に 関し,最大揚抗比をとる迎角αは実験値のものと一致 しており,揚抗比最大化という観点で本計算手法を用 いるとすれば,最も密度の低い計算格子でも利用可能 であると考えられる.後に行う最適化計算では計算格 子数が増える程全体の計算量が増大するため,必要十 分な精度を有する最低限度の格子数の使用が望ましい.

計算格子 (100×50:翼周方向×放射方向)を用い, 幾つかの NACA シリーズ翼に関して計算した結果を Fig.3 に示す (α=0). Fig.3 に示す実験値は Standard roughness のものであり, Fig.2 で用いた実験値に比べ れば翼周囲のより多くの領域が乱流場であると考えら れ,全体的により計算値に近くなっている.しかしこ の場合も実験において乱流促進がなされていないため, 翼周囲全域を乱流場とした今回の計算法を用いて,実 験状態を完全に再現することには限界があると思われ る.しかしながら,実験値が示す形状変化に対する抵 抗値の順位は計算値のものと一致しており,本計算手 法を用いて形状改良を行うことは可能であると考えら

-224-



Fig. 4 Influence of number of design parameters on solutions



Fig. 5 Influence of area constraint on solutions

NII-Electronic Library Service

-225-



Fig. 6 Convergence history in optimization procedure

れる.この結果より、後述する最適化計算では最も計 算効率が高い100 × 50計算格子を用いた.

最適化アルゴリズムには逐次線形計画法 (SLP) およ び逐次二次計画法 (SQP) を用いた.SLP アルゴリズム は濱崎ら⁷⁾の用いた逆問題解法に基づいている.本研究 で扱う問題の目的関数および制約条件は非線形である ため,まずそれらの条件式を形状変更パラメータにつ いてテーラー展開し,さらに内点法の一種であるアフィ ン変換法によって局所最適解を求め,目的関数が最小値 となるまでこの計算ステップを繰り返す.一方,SQP アルゴリズムには茨木ら⁸⁾の方法を用いた.この場合, 目的関数および制約条件の偏微分値は差分法で求めた.

3. 計算結果および考察

3.1 SLP に基づく揚抗比改良問題

前述したように、初期形状として用いたのはNACA0012 翼である.設定レイノルズ数は2,000,000とした.ま た迎角αは、NACA0012 翼の実験値の最大揚抗比迎角 の近傍として5°に固定した.まず初期形状について計 算格子(5,000 点)を楕円型方程式/代数方程式複合型 ソルバーを用いて生成し、初期粘性流場を求め、引き 続いて最適化計算を行った.目的関数は抗力/揚力比と し、これを最小化することによって揚抗比最大化問題 を解くことになる.

翼形状の変化に伴う格子再生成は、基本翼の格子情報 を用いた代数型ソルバーによって自動的に行われ、その 計算時間が全体に占める割合は無視できるほど小さい、 今回行ったケーススタディのパラメータ数、最適化繰り 返し計算回数、そして断面積一定条件の有無はTable1 に示すとおりである、共通する拘束条件は翼弦長一定、 およびY方向の形状変化が翼中心線を越えないことと した.なお(2)式を用いた場合の補間関数には区分的 線形関数を採用し,最終的に得られた形状は十分滑ら かであることを確認した.以下で計算結果を考察する.

Fig.4 に最適化過程における形状変更パラメータ数 の影響を示す. Case1 には(1) 式を用いてパラメータ 数3, Case2には(2)式でパラメータ数10, Case3には (2) 式でパラメータ数20 である.ここで示す計算結果 には面積一定拘束条件が考慮されていない. 最適化過 程が進むに従い, 揚力は常に増加傾向を示し, 抗力に関 しては Case1 以外で減少している. 揚抗比は明らかに 増加傾向にあり、本手法が有効に機能していることを示 している.またCase1以外の解は常に面積が減少して いる. Case3 の計算時間がそれ以外の Case よりはるか に多いに関わらず, 10パラメータの Case2 より揚抗比 の上昇は緩やかであり、設計パラメータの数が増える と解の収束性が低下することを示している. Case3の 場合、翼端近傍の形状変化が計算格子のねじれをもた らし、最適化過程29回で計算が発散した.これは形状 変更関数の与え方を改善することで解決できると考え られる. 解を検討した結果, 揚力および抗力の変化に 主な影響を及ぼすのは圧力成分であることがわかった.

Fig.5 には最適化過程における制約条件の影響を示 す. Case2 および Case4 共に (2) 式を用いてパラメー タ数は10 であり, Case4 の場合は面積一定条件を考慮 している. 最適化過程が進むに従い, 共に揚抗比は増 加傾向にあるが, 面積制約条件を加えた Case4 の方が やや緩やかな上昇を示す. Case4 を更に続けた結果が Case5 であり, Fig.6 はその解の収束状態を示す. 最適 化過程 50 回近傍から揚抗比の上昇は小さくなり, 最適 化過程 70 回近傍ではほぼ横這い状態に近くなる. 図に 示されるように, Case5 の面積一定条件はほぼ完全に 満たされている.

Fig.7 には今回行った各計算の最終解の表面積分値 を示す.全ての改良翼の揚抗比は基本翼のものより大 きい.また揚力および抗力変化に及ぼす影響は摩擦 応力成分よりも圧力成分の方が大きいことが分かる. 揚抗比が最も大きいのは面積一定条件を考慮しない 10 パラメータ・繰り返し 30 回 (Case2) のものである が,面積条件を考慮した10パラメータの解(Case4)も 繰り返し計算を継続することで、かなり揚抗比が改 善されている (Case5). 三角関数的分布を 3 パラメー タで用いた Casel を除き,面積条件を課さない場合 は一般に面積が減少している.一方, Case1 と Case5 を除き、翼の外周長は基本形状のものより短くなっ ている. Table1 には今回行った計算の計算時間 (DEC AlphaStation 200/4/233 を使用),および基本形状を 順問題として解いた場合を1としたときの比率が示し てある. 最適化計算時間は順問題の場合と比べ、ほぼ







Fig. 8 Comparison of solutions on wing surface $(C_p \text{ and } B)$

-227-



Fig. 9 Comparison of solutions on wing surface (frictional drag and lift distributions)



Fig. 10 Comparison of solutions on wing surface (pressure drag and lift distributions)

-228-



Fig. 11 Comparison of solutions on wing surface (δY)

パラメータ数と最適化繰り返し回数の積に比例して増 大することが分かる.これは最適化計算時間の大部分 が感度解析によって占められているためである.

Fig.8~11 において,パラメータ数10 で面積制約条 件がない場合の Case2 およびある場合の Case4 の翼表 面における解を基本形状の解と比較する.共に最適化 繰り返し数は30 回である. Case2 の揚力は Case4 の ものより大きく,また Case2 の抵抗成分は Case4 の ものより大きく,また Case2 の抵抗成分は Case4 の ものより大きく, また Case2 の抵抗成分は Case4 の ものより大きく, また Case2 の抵抗成分は Case4 の ものより大きく, また Case2 の あい. 揚抗比増加は主に揚力増加によって達成され たことが分かる.形状変更関数 B(S) の分布は Case2 の方が Pressure side(翼下面) および Suction side(翼上 面) 共に小さく,それは Y方向の形状変化量に対応し, 結果として Case2 の面積がより小さくなっている.こ こで最適化繰り返し回数が同じである Case2 と Case4 に関し,本最適化手法が如何に揚抗比を増大させたか を以下で考察する.

Fig.9 および Fig.10 には摩擦応力および圧力の揚力



Fig. 12 Comparison of convergence history between SLP (Affine) and SQP optimization methods

成分(Fl,Pl)および抵抗成分(Fd,Pd)の分布を示す. また Fig.11 には Pressure side(翼下面) および Suction side(翼上面)の,基本翼の mean-camber line を基準と したY方向の翼厚変化量(δY)を示す. 翼厚変化量は 翼下面の削り込みが大きい程より小さな負の値となり, 翼上面の張り出しが大きいほど大きな正の値となるこ とに注意されたい. 改良翼の下面の翼厚は基本翼のも のより薄く、翼の大部分において境界層外端流速が減速 され、結果としてより高い Pl分布となっている. 翼下 面の Case2 の Plが Case4 のそれより高いのは、面積一 定条件が課されていないことが影響して翼厚がより多 く削られたためであると考えられる. 翼上面の前縁近 傍を除いた大部分における Pl分布も改良翼の方が基本 翼のものより大きく、Case2、Case4 ともに基本翼より 高い揚力を生んでいることと対応している. また改良 翼の最大翼厚部は後方に移動しており、それによって翼 上面の大部分で基本翼より低い圧力分布、すなわち前 述した高い Plを生むに至っている. 翼上面の中央部近 傍までは Case4の Pl分布が Case2 のものより大きい が、表面積分の結果としては Case2 の翼下面における 揚力増加量が大きいために Case2 の揚力の方が大きい.

次に形状変化についての考察を行う.揚抗比改良翼 の揚抗比が改善されたのは抗力を極力増やさず揚力を 上げる方向へ形状変更がなされたためである.今回の 設定レイノルズ数では翼表面の前半部における大部分 が層流に近い状態と考えられるとすれば,その場合は 特に剥離を起しやすい翼上面ができるだけ流れに沿 い,境界層の発達を抑える形状が望ましい.改良翼の Suction side(翼上面)では前縁近傍で翼厚が削られてい るが,そのために基本翼で見られる急激な圧力減少が



Fig. 13 Comparison of convergence history between SLP (Affine) and SQP optimization methods

なくなっている.よってその後の圧力勾配も基本翼より 緩やかになり、これは境界層の発達をより遅らせる効 果を持つ.翼上面の中央部を含む広い領域で改良翼の Fdが基本翼より高いのは、境界層厚さが基本翼のも のより薄いことと対応している.結果的に改良翼が持 つ形状は層流翼的になった.これらの情報は形状改良 の方針を決定する際に極めて有用であると考えられる.

3.2 非線形計画法アルゴリズムの比較

ここでは非線形計画法アルゴリズムの比較を SLP と SQP で行った結果について述べる.Fig.12 において, 上述した SLP の解 (Case5) と,最適化アルゴリズムを SQP に変更した場合の解 (SQP)の収束履歴を比較す る.SQP による解は最適化繰り返し数 15 回において 設計変数の変動量が小さくなり (10⁻⁸以下),収束と見 なして計算を終了した.Fig.13 は Case5 および SQP の最終翼形状である.若干の差はあるものの,両者は ほぼ同様な方向へ改良されていると思われる.SQP に 基づく手法の計算時間は SLP 場合の約 2.5 倍であった が,収束性に着目すればより有効であると考えられる.

4. 結言

本研究では RaNS 方程式の数値解に基づく二次元翼 の揚抗比改良問題を解いた.本研究の結論をまとめる と以下のとおりである.

- 1. RaNS 方程式法を非線形計画法と結合して二次元 翼形状改良問題を解いた結果,従来非粘性流法で は評価できなかった粘性抵抗成分を考慮した形状 最適化が可能となった.
- 2. 最適化解を検討することにより, 翼形状設計にお ける改良方針を提供できることが示された.
- 3. 非線形計画法のアルゴリズムとして SLP と SQP を用いた結果,ほぼ同等な解を得ることが示された. SQP を用いた場合の計算時間は SLP の場合

の数倍であるが、収束性は優れていることが明ら かとなった.

本研究では一定迎角における揚抗比向上を目指した 翼形状改良を試みた.しかし実際にはモーメントや揚 力中心の移動なども考慮する必要があると考えられる が,それらは今後の課題としたい.

なお本研究の一部は(社)日本造船研究協会との共同 研究 (SR229 研究部会)として行われました.委員各位 の御助言に謝意を表します.

参考文献

- 日夏宗彦,日野孝則:船体周りの流れの計算法, 船体周りの流れと船型開発に関するシンポジウム, April 1993, pp.225-262
- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994
- 日本造船研究協会:第222研究部会「大型肥大船 船尾流場推定法の高度化」成果報告書,1996
- 4) 鈴木和夫,小柴幸雄:船型計画法と線図生成,船 体周りの流れと船型開発に関するシンポジウム, April 1993, pp.163-201
- Tahara, Y.: Computation of Viscous Flow around Series 60 Model and Comparison with Experiments, J. Kansai Society of Naval Architects, Japan, No. 220, September 1993, pp. 29-47
- Abbott, H., and Von Doenhoff, A.E.: Theory of Wing Sections, Dover Publications, 1959
- 7) 浜崎準一,姫野洋司,田原裕介:非線形計画法に よる船型最適化の試み(第4報)一伴流と粘性抵抗 を考慮した船体後半部形状の改良,関西造船協会 誌,第226号,1996, pp.15-21
- 8) 茨木俊秀,福島雅夫:FORTRAN77 最適化プログラミング,岩波コンピュータサイエンス,1991, pp. 167-207

論

[討論] (横浜国立大学工学部)池畑光尚

討

翼形状については従来理想流体中のポテンシャル流 れの理論で研究され,実用にも十分耐える立派な成果 が得られて今日の航空機や舶用プロペラの発達がもた らされたと思います.そのような現状の中でNSソル バーを使用して二次元翼形状改良問題を解かれたのに は,何か特別な意図あるいは理由があるのでしょうか. お考えをお聞かせ下さい. [回答]

ご指摘のとおり、ポテンシャル理論による翼形状改 良問題に関しては多くの研究があります.したがって、 単純な翼形状の最適化問題に RaNS 法を用いる必要は ないのでありますが、本研究は三次元船体の形状最適 化問題への適用の第一段階として、著者の RaNS コー ドと最適化手法の結合の妥当性を検討するための事例 として取り上げたのが理由の一つであります.また本 研究では RaNS 方程式を解くことにより、従来ポテン シャル理論もしくは境界層理論で困難とされてきた諸 問題を克服し、より実際的な流場情報に基づく形状最 適化も目指しています.貴重なご指摘ありがとうござ いました.

時間平均速度勾配テンソルの第2不変量を用いた k-ε 乱流モデルの再構成に関する研究 *1

学生会員沖本憲司*2,正会員姫野洋司*3,正会員田原裕介*3

Improvement of $k-\epsilon$ Turbulence Models with Second Invariant of Gradient of Time-Averaged Velocity Tensor

By Kenji OKIMOTO (Student Member), Yoji HIMENO (Member) and Yusuke TAHARA (Member)

Many numerical schemes have been developed to reproduce complicated flow fields around ship stern. However, most of them have turned out to be insufficient due to inadequate accuracy of the numerical scheme, grid resolution, and turbulence modeling. The recent workshops⁷ stated that turbulence modeling may be the most important issue for the problem. This paper presents a new turbulent energy dissipation model for application to a turbulent flow with longitudinal vortex. The standard ϵ equation is modified so as to provide more realistic time mean flow and Reynoldsstress fields. The fully-elliptic RaNS (Reynolds-averaged Navier-Stokes equation) and continuity equations are solved with an eddy-viscosity turbulence model, where k and ϵ are solved using twolayer technique. Discussions are made concerning the validity of the new turbulence model comparing with existing turbulence models and experiments.

Keywords : Turbulence Models, $k \cdot \epsilon$ Model, Longitudinal Vortex, Reynolds-Stress, Flat Plate, Dissipation Equation

1. 緒言

近年コンピューターの発達に伴い、乱流の実用的な 解析法の必要性が望まれ、そのために乱流モデルが 考案されるようになった。そのなかでも工学的乱流モ デルでは、レイノルズ応力のような乱流統計量を予 測するため、さまざまなモデルや概念が提案されて きた。Reynolds は時間平均をほどこしたナビエ・ス トークス方程式の概念を提案し、その方程式を閉じる 方法として Boussinesq は渦動粘性係数の考えを用い た。Prandtl は混合距離の概念を用いて渦動粘性係数 を乱流エネルギー k と結び付け、k 輸送方程式を与え た。Davidov¹⁾によって導入されたエネルギー消散率 ϵ 輸送方程式の概念は Hanjalic²⁾などの研究によって 一層の進展をみた。これらの研究をもとに、高レイ

*3 大阪府立大学工学部

ノルズ数流れに対する標準 k- ϵ モデルが Launder and Spalding³⁾ によって与えられ,同モデルは現在もなお 広く利用されている.さらに Shih and Lumley⁴⁾によっ て,構成方程式の手法を利用した ϵ 消散方程式の生産 項・消散項のモデリングが行われ,これは現在の乱流 モデルの基本表現となっている.しかし,このモデル においても,船尾流に特徴的な旋回,回転効果等の渦 を伴う乱流場では,信頼度が落ちるといわれている⁷⁾.

本研究の目的は、この問題を解決するため、k-e2方 程式モデルにおいて、特に eの消散方程式を再構成す ることにより、従来のモデルでは得られなかった縦渦 を含む乱流場の予測精度を改良することである.

2. 消散方程式の再構成

本モデルの特徴は、乱流消散方程式に時間平均速度 場の渦度テンソルと歪みテンソルからなる第2不変量 を新たに導入することにより再構成している点である. 第2不変量の導入により、従来の消散方程式では得ら

^{*1} 平成 10 年 5 月 21 日関西造船協会春季講演会において 講演,原稿受付平成 10 年 6 月 10 日

^{*2} 大阪府立大学大学院

れなかった強い縦渦を含む乱流場を精度よく表現する ことが可能となる.

まず標準型 k-ε モデルをテンソル表記で以下に示す.

$$\frac{\partial k}{\partial t} + \overline{U_j} \frac{\partial k}{\partial x_j} = P_k - \epsilon + \frac{\partial}{\partial x_j} \left(\frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right)$$
(2.1)

$$\frac{\partial \epsilon}{\partial t} + \overline{U_j} \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} (\frac{\nu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j}) + 2C_{\epsilon 1} C_\mu \overline{S}_{ij} \overline{S}_{ij} k$$

$$-C_{\epsilon 2} \frac{\epsilon^2}{k}$$
 (2.2)

 $\sigma_k, \sigma_\epsilon$ は乱流プラントル数, $C_{\epsilon 1}, C_{\epsilon 2}$ はモデル定数 である.ここで, (2.1) 式の右辺はそれぞれ順に生産, 消散, 拡散を表す. 同様に (2.2) 式の右辺はそれぞれ順 に拡散, 生産, 消散を表す. (2.2) 式の右辺第2項と第 3項をとりだすと, 次式のように表すことができる.

$$P_{\epsilon} - \Phi_{\epsilon} = C_{\epsilon 1} \nu_t \frac{\partial \overline{U_i}}{\partial x_j} (\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i}) \frac{\epsilon}{k} - C_{\epsilon 2} \frac{\epsilon^2}{k} \quad (2.3)$$

(2.3) 式を見れば分かるように、平均速度勾配からな る右辺第1項の2つの成分 $\frac{\partial U_i}{\partial x_j} \partial \frac{\partial U_i}{\partial x_j} e^{\frac{\partial U_i}{\partial x_j}} e^{\frac{\partial U_i}{\partial x_j}} e^{\frac{\partial U_i}{\partial x_j}} e^{\frac{\partial U_i}{\partial x_j}} e^{\frac{\partial U_i}{\partial x_i}} e^{\frac{\partial U$

$$P_{\epsilon} - \Phi_{\epsilon} = C_{\epsilon 11} \nu_t \frac{\partial \sigma_l}{\partial x_j} \frac{\partial \sigma_l}{\partial x_j} \frac{e}{k}$$

$$+ C_{\epsilon 12} \nu_t \frac{\partial \overline{U_i}}{\partial x_i} \frac{\partial \overline{U_j}}{\partial x_i} \frac{\epsilon}{k} - C_{\epsilon 2} \frac{\epsilon^2}{k}$$
(2.4)

 $C_{\epsilon 11}, C_{\epsilon 12}, C_{\epsilon 2}$ はモデル定数である. (2.4)式を平均速度歪みテンソルと平均速度渦度テンソルで展開することにより,新しい ϵ 輸送方程式は次のように変形される.

$$\frac{\partial \epsilon}{\partial t} + \overline{U_j} \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + 2C_{\epsilon 1} C_\mu \overline{S}_{ij} \overline{S}_{ij} k + 2C_{\overline{q}} C_\mu \overline{Q} k - C_{\epsilon 2} \frac{\epsilon^2}{k}$$
(2.5)

ただし,

$$\overline{Q} = \frac{1}{2} (\overline{\Omega}_{ij} \overline{\Omega}_{ij} - \overline{S}_{ij} \overline{S}_{ij})$$
(2.6)

 $C_{\overline{q}}$ はモデル定数である. \overline{Q} は平均速度勾配テンソルの座標系に依存しない第2不変量であり、流体の回転(平均速度渦度テンソル)と歪み(平均速度歪みテンソル)の差で表現される. \overline{Q} が正の値をとる場合, $\overline{\Omega}_{ij}\overline{\Omega}_{ij}$ が $\overline{S}_{ij}\overline{S}_{ij}$ を上回っている.局所的な散逸

が $2\nu \overline{S}_{ij}\overline{S}_{ij}$ と表されるので、この場合は、散逸より もむしろ回転流、すなわち渦核を伴う流場を表してい る.上式 (2.5) で右辺はそれぞれ順に拡散、生産、第2 不変量を伴う生産、消散を表す、平均速度勾配からな る2つの成分を分離することにより、平均速度歪みテ ンソルのみならず平均速度渦度テンソルの影響も表現 することが可能となることが期待される.

(2.1), (2.2) 式が従来の標準型 $k - \epsilon = \tau \nu$,(2.1), (2.5) 式が新しい $k - \epsilon = \tau \mu \tau \sigma \delta \delta$. kの輸送方程式について は,従来のものと変更はないが, ϵ については (2.2), (2.5) 式からわかるように, (2.4) 式を平均速度歪みテ ンソルと平均速度渦度テンソルで展開することによ り,従来の ϵ 輸送方程式 (2.2) に新しい項 $2C_q C_\mu Q k \pi \sigma$ 加わった形となっている.

この新たに加わった項, $2C_{\overline{q}}C_{\mu}\overline{Q}k$ は, 渦度に関連す る反対称成分の影響を表しており, この第2不変量 \overline{Q} により, 縦渦を評価できる.また,境界層流れにおい てもこの項は,ほぼ0となり従来のモデルに帰着する という利点もある.以下では, (2.5)式の係数 $C_{\overline{q}}$ の値 について検討する.

2.1 *ϵ* 方程式の係数 *C_a* の選定

新しい ε 消散方程式のモデル化において,従来と 同様に 2 次元乱流を考える.ここで局所平衡を仮定し (2.5)式を書き換えると,

$$\frac{\partial \epsilon}{\partial t} + \overline{U_j} \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + 2C'_{\epsilon 1} C_\mu \overline{S}_{ij} \overline{S}_{ij} k - C'_{\epsilon 2} \frac{\epsilon^2}{k}$$
(2.7)

ただし,

$$C'_{\epsilon 1} = C_{\epsilon 1} - \frac{1}{2}C_{\overline{q}} \qquad C'_{\epsilon 2} = C_{\epsilon 2} - \frac{1}{2}C_{\overline{q}} \qquad (2.8)$$

式 (2.7) は、この場合には従来の消散方程式に帰着 することが分かる.ここで従来から行われてきた消散 方程式のモデル化を行う⁴⁾.

まず格子背後の減衰乱流を考える.この流れ場では kの輸送方程式と ε の輸送方程式はそれぞれ次のよう に表される.

$$\frac{\partial k}{\partial t} = -\epsilon \tag{2.9}$$

$$\frac{\partial \epsilon}{\partial t} = -C_{\epsilon 2}' \frac{\epsilon^2}{k} \tag{2.10}$$

式(2.9)と式(2.10)より,

$$\frac{\partial^2 k}{\partial t^2} = C'_{\epsilon 2} \frac{1}{k} \left(\frac{\partial k}{\partial t}\right)^2 \tag{2.11}$$

が導かれる.格子背後の減衰乱流では,減衰初期に次の関係が成立することが実験的に確認されている.

NII-Electronic Library Service

$$k \propto t^{-1} \tag{2.12}$$

式 (2.12) を式 (2.11) に代入すれば, $C'_{c2} = 2.0$ が得られる.次に圧力勾配のない定常な 2 次元壁乱流 (x_2 方向が壁法線方向)を考える.このような流れ場においては, $D\epsilon/Dt = 0$ と,局所平衡を仮定する.したがって式 (2.7) は,

$$\frac{\epsilon^2}{k}(C'_{\epsilon_1} - C'_{\epsilon_2}) + \frac{\partial}{\partial x_2}(\frac{\nu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_2}) = 0$$
 (2.13)

となる.ここで,kは x_2 方向に一定であると仮定した.式 (2.13) に壁乱流における ϵ の関係式,

$$\epsilon = \frac{C_{\mu}^{3/4} k^{3/2}}{\kappa x_2} \tag{2.14}$$

を代入すれば、

$$C_{\epsilon 1}' - C_{\epsilon 2}' = -\frac{\kappa^2}{\sigma_{\epsilon} C_{\mu}^{1/2}}$$
(2.15)

以上から,

$$C'_{\epsilon 1} = 1.55$$
 $C'_{\epsilon 2} = 2.0$ (2.16)

を得る.これが従来, $k-\epsilon$ モデルの理論値と称されている値である.

一方, Launder and Spalding³⁾は、各種計算から2 次元乱流境界層の実験値に合うようモデル定数を fittingし,

$$C'_{\epsilon 1} = 1.44$$
 $C'_{\epsilon 2} = 1.92$ (2.17)

とした. これがいわゆる k- ϵ モデルの標準値とされている. ここで,代表的な k- ϵ モデルのモデル定数を Table 1 に示す.

Table 1 Summary of model constants.

~····		,		r
Models	$C_{\epsilon 1}$	$C_{\epsilon 2}$	σ_k	σ_{ϵ}
Standard(1974)	1.44	1.92	1.0	1.3
Jones-Launder(1972)	1.55	2.0	1.0	1.3
Launder-Sharma(1974)	1.44	1.92	1.0	1.3
Hoffmann(1975)	1.81	2.0	2.0	3.0
Reynolds(1976)	1.0	1.83	1.69	1.3
Hassid-Poreh(1978)	1.45	2.0	1.0	1.3
Lam-Bremhorst(1981)	1.44	1.92	1.0	1.3
Chen(1982)	1.35	1.8	1.0	1.3
To-Humphrey(1986)	1.44	1.92	1.0	1.3
Nagano-Hishida(1987)	1.45	1.9	1.0	1.3
Myong-Kasagi(1990)	1.4	1.8	1.4	1.3
Nagano-Togawa(1990)	1.45	1.9	1.4	1.3
Shih-Mansour(1990)	1.45	2.0	1.3	1.3
Michelassi-Rodi(1991)	1.44	1.92	1.3	1.3
Abe-Nagano(1992)	1.5	1.9	1.4	1.4
Yang-Shih(1993)	1.44	1.92	1.0	1.3

Table 1 から,式 (2.16) は Jones-Launder⁵⁾の理論値 であり、またこの理論値と他の $C_{\epsilon 1}$, $C_{\epsilon 2}$ の値を比較す ると、ほぼ等しい差で減っていることが分かる。例え ば、Jones-Launder⁵⁾と'Standard'とを比較すると、 'Standard'は理論値である Jones-Launder⁵⁾に比べて $C_{\epsilon 1}$, $C_{\epsilon 2}$ ともに約 0.1づつ減少していることが分かる。

また Jones-Launder⁵⁾と Chen⁶⁾とを比較すると, C_{c1} , C_{c2} ともに 0.2 づつ減少していることが分かる. このこ とは,式 (2.8) より説明できる. すなわち式 (2.8) を, 前述のような Jones-Launder⁵⁾と 'Standard 'あるい は Jones-Launder⁵⁾と Chen⁶⁾のモデル定数を比較した 式であると考えると,その差はともに式 (2.8) から $\frac{1}{2}C_{\overline{q}}$ となっている.

したがって、Jones-Launder⁵⁾と 'Standard 'のモ デル定数を比較すると,式(2.8)からC'_1, C'_2が 'Standard 'の値, C_{ϵ_1} , C_{ϵ_2} が Jones-Launder⁵⁾の値の とき $C_{\overline{q}} = 0.2$ と考える事ができる. あるいはまた C'_{e1} , $C'_{\epsilon 2}$ が Jones-Launder⁵⁾の値, $C_{\epsilon 1}$, $C_{\epsilon 2}$ が 'Standard ' の値のとき $C_{\overline{q}} = -0.2$ とみなすこともできる. 同様に Jones-Launder⁵⁾と Chen⁶⁾のモデル定数の比較を考え ると,式(2.8)から $C'_{\epsilon 1}$, $C'_{\epsilon 2}$ がChen⁶⁾の値, $C_{\epsilon 1}$, $C_{\epsilon 2}$ が Jones-Launder⁵⁾の値のとき $C_{\overline{q}} = 0.4$ となり, C'_{ϵ_1} , C_{ϵ_2}' が Jones-Launder⁵⁾の値, C_{ϵ_1} , C_{ϵ_2} が Chen⁶⁾の値の とき $C_{\overline{q}} = -0.4$ となる.いずれの組み合わせが妥当 であるかは、この枠組みでは判断できないので、こ こでは, Jones-Launder⁵⁾と 'Standard 'モデルが平 板境界層流れについての数値計算で信頼性が高いと 考え,前者の $C_{\overline{q}} = 0.2$ のケースをmod.とし,後者の $C_{\overline{q}} = -0.2$ のケースを mod.1 として扱う.

2.2 *ϵ* 方程式の係数について

新しい ϵ 消散方程式 (2.7) において 2 つの解釈をした. つまり式 (2.7) は従来の消散方程式に帰着しているので、従来にならい $C'_{\epsilon1}$, $C'_{\epsilon2}$ に'Standard'の値を用いるという考え方 (mod.) と、新しい消散方程式 (2.5) は従来にはない平均速度渦度テンソルの影響が含まれているので、従来のように $C_{\epsilon1}$, $C_{\epsilon2}$ に'Standard'の値を用い、渦度テンソルの影響を生かす (mod.1) という 2 つの考え方である.

前者が C'_{ϵ_1} , C'_{ϵ_2} に 'Standard 'の値を用いること から mod., 後者が C'_{ϵ_1} , C'_{ϵ_2} に Jones-Launder⁵⁾の理論 値を用いることから mod.1 とおいたものである.

ケース mod. では $C_{c1} = 1.55$, $C_{c2} = 2.0$, $C_{\overline{q}} = 0.2$ であり、これは式 (2.5)の右辺第2項、第3項に着目す ると、右辺第2項の平均速度歪みテンソルにかかるモ デル定数が 1.55 で第3項の $C_{\overline{q}}$ が正かつ第2不変量の なかで平均速度歪みテンソルは負であるから、全体と して右辺第2項の平均速度歪みテンソルにかかるモデ ル定数を減らす方向に働くことがわかる.その結果右 辺第2項の平均速度歪みテンソルにかかるモデル定数 が 1.44 となり'Standard'モデルに帰着したものと なる. また $k や \epsilon$ については,結果として両方とも従 来の $k - \epsilon$ モデルでの値よりも増大したものとなる.

ケース mod.1 では $C_{\epsilon 1} = 1.44$, $C_{\epsilon 2} = 1.92$, $C_{\overline{q}} = -0.2$ であり、これは式 (2.5)の右辺第2項、第3項に 着目すると、右辺第2項の平均速度歪みテンソルにか かるモデル定数が 1.44 で第3項の $C_{\overline{q}}$ が負、かつ第2 不変量のなかで平均速度歪みテンソルは負であるから 全体として右辺第2項の平均速度歪みテンソルにかか るモデル定数を増やす方向に働くことがわかる.その 結果右辺第2項の平均速度歪みテンソルにかかるモデ ル定数が 1.55 となり Jones-Launder⁵⁾モデルに帰着し たものとなる.また $k \approx \epsilon$ については、ケース mod. と同様に結果として両方とも従来の $k - \epsilon$ モデルでの 値よりも増大する.

以下では、これらの係数を用いた計算を実施し、実 験値と比較検討することにする.

3. 支配方程式と乱流モデル

本モデルの有効性を検証するため,平板境界層内の 縦渦流れについて計算を実施した.

まず、3次元、非定常、非圧縮流れに関する直交座 標系における運動方程式を考える.無次元系における レイノルズ平均ナビエストークス方程式と連続の式は それぞれ次のようになる.

$$\frac{\partial \overline{U_i}}{\partial x^i} = 0 \tag{3.18}$$

$$\frac{\partial \overline{U_i}}{\partial t} + \overline{U_j} \frac{\partial \overline{U_i}}{\partial x^j} + \frac{\partial \overline{u_i u_j}}{\partial x^j} + \frac{\partial p}{\partial x^i} - \frac{1}{R_e} \nabla^2 \overline{U_i} = 0 \quad (3.19)$$

$$z z \mathcal{T},$$

$$\nabla^2 = \frac{\partial^2}{\partial x^j \partial x^j} \tag{3.20}$$

-235-

またここで, $\overline{U_i} = (\overline{U}, \overline{V}, \overline{W}), u_i = (u, v, w)$ はそれ ぞれ平均部分と変動部分を表しており, 代表速度 U_0 で 無次元化される. $x^i = (x, y, z)$ は代表長さLによって 無次元化され, 圧力 p は ρU_0^2 で無次元化される. R_e は レイノルズ数である.

またレイノルズ応力を平均速度勾配と結び付け次式 のように与えられる.

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial \overline{U_i}}{\partial x^j} + \frac{\partial \overline{U_j}}{\partial x^i}\right) - \frac{2}{3}\delta_{ij}k \tag{3.21}$$

ここでkは乱流エネルギー, δ_{ij} はクロネッカーのデ ルタを表す. (3.21)を (3.19)に代入すると次式がえら れる.

$$\frac{\partial \overline{U_i}}{\partial t} + ((\overline{U_j} - \frac{\partial \nu_t}{\partial x^j}) \frac{\partial \overline{U_i}}{\partial x^j}$$

$$-\frac{\partial\nu_t}{\partial x^j}\frac{\partial\overline{U_j}}{\partial x^i}) + \frac{\partial}{\partial x^i}(p+\frac{2}{3}k) - \frac{1}{R_{\phi}}\nabla^2\overline{U_i} = 0 \quad (3.22)$$

ここで、 $\frac{1}{R_{\phi}} = \frac{1}{R_{e}} + \nu_{t}, \phi = \overline{U_{i}}$ (*i* = 1, 2, 3) 渦粘性 ν_{t} が与えられると、式 (3.18) と式 (3.22) は

渦粘性 ν_t が与えられると,式 (3.18)と式 (3.22)は $\overline{U_i},p$ に関して解かれる.2方程式モデルにおいては, 渦粘性 ν_t を乱流エネルギー kと消散率 ϵ と関連付け次 式のように与えられる.

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \tag{3.23}$$

またここで従来の'Standard '形の $k - \epsilon$ 輸送方程 式は次のようになる.

$$\frac{\partial k}{\partial t} + (\overline{U_j} - \frac{1}{\sigma_k} \frac{\partial \nu_t}{\partial x^j}) \frac{\partial k}{\partial x^j} - \frac{1}{R_k} \nabla^2 k - G + \epsilon = 0$$
(3.24)

$$\frac{\partial \epsilon}{\partial t} + (\overline{U_j} - \frac{1}{\sigma_{\epsilon}} \frac{\partial \nu_t}{\partial x^j}) \frac{\partial \epsilon}{\partial x^j} - \frac{1}{R_{\epsilon}} \nabla^2 \epsilon$$
$$- C_{\epsilon 1} \frac{\epsilon}{k} G + C_{\epsilon 2} \frac{\epsilon^2}{k} = 0 \qquad (3.25)$$

ここで, $\frac{1}{R_{\phi}} = \frac{1}{R_{e}} + \frac{\nu_{t}}{\sigma_{\phi}}$, $\phi = k, \epsilon$. また, *G*は生産率であり以下のように表される.

$$G = \frac{1}{2}\nu_t (\frac{\partial \overline{U_i}}{\partial x^j} + \frac{\partial \overline{U_j}}{\partial x^i})^2$$
(3.26)

そして, $(C_{\mu}, C_{\epsilon 1}, C_{\epsilon 2}, \sigma_k, \sigma_{\epsilon})$ は定数であり,その値 はそれぞれ (0.09,1.44,1.92,1.0,1.3)である.本研究の 改良形では式 (3.25)のかわりに式 (2.5)を用いる.

壁近傍においては、 ν_t は輸送方程式 (3.24) から得ら れる k によってのみ決められ、 $\nu_t = C_\mu \sqrt{k} l_\mu$ のように 表される.そこでは、消散率 ϵ は、輸送方程式 (3.25) を解かず、k によって表され、 $\epsilon = \frac{k^{3/2}}{l_\epsilon}$ と指定される. ただし、 $\epsilon = \frac{k^{3/2}}{l_\epsilon}, l_\epsilon = C_l y [1 - \exp(-R_y/A_\epsilon)]$ である.

これらの長さスケールは乱流レイノルズ数 $R_y = R_c \sqrt{ky}$ の項によってダンピングの効果が含まれている.また yは 壁からの距離を表す. Chen and Patel⁶⁾に従うと,定数 C_l, A_μ, A_ϵ はそれぞれ, $C_l = \kappa C_\mu^{-3/4}, A_\mu = 70, A_\epsilon = 2C_l$ と与えられる.ここで, κ はカルマン定数 (=0.418) で ある. C_l の値は内層と外層の境界でなめらかな渦動粘 性分布が得られるように決められている. A_μ は平板境 界層の場合における対数則で付加定数 B = 5.45 を表 すように決められている. A_c は消散率 ϵ の漸近挙動が 得られるように決められている.

3.1 支配方程式の離散化と速度-圧力カップリング

5つの輸送方程式(3.22),(3.24),(3.25)は有限解析ス キームによって離散化される.スキームにおいて,方程 式は各々局所数値要素で線形化され,変数分離によっ て解析的に解かれる.内部格子点での解析解を評価す ることにより,12点離散式が得られ次のようになる.

 $\phi_p = \frac{1}{1 + C_p (C_U + C_D + \frac{R}{\Delta \tau})} [C_{NE} \phi_{NE} + C_{NW} \phi_{NW}]$

 $+C_{SE}\phi_{SE}+C_{SW}\phi_{SW}+C_{EC}\phi_{EC}+C_{WC}\phi_{WC}+C_{NC}\phi_{NC}$

$$+C_{SC}\phi_{SC} + C_p(C_U\phi_U + C_D\phi_D + \frac{R}{\Delta\tau}\phi_p^{n-1}) - C_p(S_{\phi})_p]$$

$$(3.27)$$

式 (3.27)の有限解析係数を計算し、乱流量 k, ϵ について解く. それから 2 層乱流モデルを用いて渦動粘 性分布を更新させ、速度場に関する運動方程式と圧力 方程式が連続の式を満足するように解かれる.これら 一連の解法は田原⁷⁾⁸⁾⁹⁾に基づいている.

3.2 境界条件及び初期条件



Fig. 1 Computational grid

Fig.1 のように下面 (y = 0) を平板としたものを考える. 座標は直交座標であり, x 軸と z軸が水平面上に y 軸は鉛直上向きを正にとる.

計算領域は Fig.2 のようになる.ここで、 S_i, S_e はそれ ぞれ流入と流出、 S_b は壁面境界、 S_t は上部境界、 S_{s1}, S_{s2} は側面を表す.境界条件は S_b 上で $\overline{U} = \overline{V} = \overline{W} = 0, S_t$ 上で $\overline{U} = 1, \overline{V} = \overline{W} = 0.$

流入境界条件は $S_i \pm (x=2550 \text{ mm})$ で Suzuki¹⁰⁾らに よる大阪大学風洞での実験値を用いることとする.しか し, ϵ 分布については,実験値がないため Boussinesq の式 (3.21) と ν_i の定義式 (3.23) から ϵ を算出し初 期値とした.なお, $S_i \pm$ で式 (3.21) の平均速度勾配 $\frac{\partial \overline{U_i}}{\partial x^i}$, $\frac{\partial \overline{U_j}}{\partial x^i}$ を求める際には,x軸方向に1次精度片側 差分を,y,z軸方向に2次精度中心差分をとった.な お,繰り返し数5回以後は領域内の ϵ を用い,外層法 により $S_i \pm$ の ϵ を更新した.平板の長さは4000mmで あり,翼がx=2400 mmの位置にとりつけられており, そこから縦渦が発生している.計測面は7断面あり, x=2550,2625,2700,2775,2850,2925,3000 mmである.代 表長さは3000 mmであり, ν イノルズ数は3,664,500 である.計算ではこの7断面を代表長さ3000 mmで 割った無次元長さを用いる.

4. 計算結果および考察

計算結果について, 値は長さ*L*, 速度*U*, 密度ρなど で無次元化される.

計算グリッドの x 軸では,流入面と流出面はそ れぞれ x=(0.85,1.05), y軸では y=(0,0.06), z軸では z=(-0.06,0.06) となっている.グリッド数は 100 × 40 × 30(=120,000) であり,レイノルズ数は実験と 同じ 3,664,500 である.時間刻み $\Delta \tau$ と速度緩和係 数 ($\alpha_u, \alpha_v, \alpha_w$),乱流量 (α_k, α_c),圧力 (α_p)はそれ ぞれ次のようになる. $\Delta \tau = 0.01$, ($\alpha_u, \alpha_v, \alpha_w$) = (1,1,1),(α_k, α_c) = (0.01,0.01), $\alpha_p = 0.1$ である.





標準型 $k-\epsilon$ モデル,新しい $k-\epsilon$ モデル mod.,mod.1の

-236-

3 種類の数値計算結果と実験値を Fig.3 から Fig.11 に 示す. なお等値線図はすべて,7断面のうちの最終断 面での値を示す.

Fig.3 は時間平均速度の主流方向成分 \overline{U} の等値線図 である.実験値と比較して,標準型 $k-\epsilon$ モデル,新しい $k-\epsilon$ モデル mod.および mod.1 はどれもほぼ同じ等値線 図が得られている.特に新しい $k-\epsilon$ モデル mod.,mod.1 では,渦核付近での右まわりの境界層ねじれがよく捉 えられている.

Fig.4 は時間平均速度成分 \overline{V} の等値線図である.時間平均速度成分 \overline{V} は渦度の主流方向成分 Ω_x に関連する時間平均速度成分であり、これについては実験値と比較して、標準型 k- ϵ モデルは渦核付近でやや減衰しているのに対し、新しい k- ϵ モデル mod.,mod.1 は実験値とほぼ同じ等値線図が得られている.これは、新しい k- ϵ モデルでは、モデル化において平均速度歪みテンソルのみならず平均速度渦度テンソルの影響も考慮されているのに対し、標準型 k- ϵ モデルでは、平均速度渦度テンソルの影響が考慮されていないためである.

Fig.5 は時間平均速度成分 \overline{W} の等値線図である.時間平均速度成分 \overline{W} も渦度の主流方向成分 Ω_x に関連する時間平均速度成分である.実験値と比較して,標準型 $k-\epsilon$ モデルは渦核付近でやや減衰しているのに対し,新しい $k-\epsilon$ モデル mod.,mod.1 は実験値とほぼ同じ等値線図が得られている.これも Fig.4 と同様に,新しい $k-\epsilon$ モデルでは,モデル化において平均速度渦度テンソルの影響も考慮されているのに対し,標準型 $k-\epsilon$ モデルでは,平均速度渦度テンソルの影響が考慮されていないためである.

Fig.6 はレイノルズ応力 \overline{wv} の等値線図である.実験 値と比較して、どのモデルもほぼ同じ等値線図が得ら れている.Fig.7 はレイノルズ応力 \overline{vw} の等値線図であ る.レイノルズ応力 \overline{vw} は渦度の主流方向成分 Ω_x に 関連するレイノルズ応力であり、これについては標準 型 $k-\epsilon$ モデルは渦核付近で減衰しているのに対し、新 しい $k-\epsilon$ モデル mod.,mod.1 は実験値とほぼ同じ等値 線図が得られている.これも Fig.5 と同様に、新しい $k-\epsilon$ モデルでは、モデル化において平均速度渦度テン ソルの影響も考慮されているのに対し、標準型 $k-\epsilon$ モ デルでは、その影響が考慮されていないためである.

Fig.8 はレイノルズ応力 \overline{uw} の等値線図である.実験値と比較して、どのモデルもほぼ同じ等値線図が得られている.Fig.9 は渦度の主流方向成分 Ω_x の等値線図である.実験値と比較して、標準型 k- ϵ モデルは渦核付近でかなり減衰しているのに対し、新しい k- ϵ モデル mod.,mod.1 は実験値とほぼ同じ等値線図が得られている.これも Fig.7 と同様に、新しい k- ϵ モデルでは、平均速度渦度テンソルの影響がも考慮されてい

るためである.

Fig.10は乱流エネルギー k の等値線図である.実験 値と比較して、どのモデルも渦核付近での強い等値線 図が得られておらず、この原因としては等方性の概念 のもとでの Boussinesq 式の限界が考えられる.Fig.11 は消散率 ϵ の等値線図である.標準型k- ϵ モデル、新 しいk- ϵ モデル mod.および mod.1ともほぼ同様な等 値線図が得られている.

また z方向での時間平均速度分布 $\overline{U}, \overline{V}, \overline{W}, \nu イノル$ ズ応力分布 $\overline{uv}, \overline{vw}, \overline{uw}, \overline{a}$ 度分布 Ω_x , 乱流エネルギー分 布 k, 消散率分布 ϵ をそれぞれ Fig.12 から Fig.20 に示 す. ここで y は渦核での y の値である. Fig.12 の平均速 度 \overline{U} ではどのモデルにおいてもほぼ同じ分布が得られて おり,実験値を捉えた挙動が得られている. Fig.13 の平 均速度 \overline{V} では渦度が支配的な領域-0.015 < z < 0.002 において標準型 k- ϵ モデルはかなり減衰しているのに 対し,新しい k- ϵ モデル mod.,mod.1 は実験値とほぼ 同じ分布が得られている. それ以外の領域ではどのモ デルも実験値とほぼ同じ分布が得られている. これも 平均速度渦度テンソルの影響が考慮されているためで ある. Fig.14 の平均速度 \overline{W} に関しても標準型 k- ϵ モデル mod.,mod.1 は実験値を捉えた挙動が得られている.

Fig.15 のレイノルズ応力 \overline{wv} ではローカルな領域 -0.03 < z < -0.02 において標準型 k- ϵ モデルよりも 新しい k- ϵ モデル mod.,mod.1 の方が精度が悪くなっ ている.これは,新しい k- ϵ モデルにおいてそのロー カルな領域で乱流エネルギー k が過大評価され,その 結果レイノルズ応力を過小評価しているためである. Fig.16 のレイノルズ応力 \overline{wv} では渦度が支配的な領域 -0.015 < z < 0.002 において標準型 k- ϵ モデルはかな り減衰しているのに対し,新しい k- ϵ モデルでは mod.1 で少しばらつきはあるものの,実験値とほぼ同じ分布 が得られている.それ以外の領域ではどのモデルも実 験値とほぼ同じ分布が得られている.Fig.17 のレイノ ルズ応力 \overline{uw} ではどのモデルにおいても少しばらつき はあるものの,実験値を捉えた挙動が得られている.

Fig.18の流れ方向渦度 Ω_x では渦度が支配的な領域 -0.015 < z < 0.002 において標準型 k- ϵ モデルはかなり 減衰しているのに対し,新しい k- ϵ モデル mod.,mod.1 は実験値とほぼ同じ分布が得られている.それ以外の 領域ではどのモデルも実験値とほぼ同じ分布が得られ ている.これも Fig.16 と同様に新しい k- ϵ モデルでは, 平均速度渦度テンソルの影響も考慮したためである. 一方, Fig.19 の乱流エネルギー k では渦度が支配的な 領域-0.015 < z < 0.002 において, どのモデルも実験 値の強い分布が得られていない.これは前述のように, 等方性の概念のもとでの Boussinesq 式の限界が考えら

-237-

れる. Fig.20の消散率 ϵ では標準型 k- ϵ モデルと比較 して,新しい k- ϵ モデル mod.,mod.1 とも一様に高い分 布が得られている. これは,新しい ϵ 消散方程式にお いて第2不変量および乱流エネルギー kの変動の結 果, ϵ が増加したと考えられる.

以上の検討により、レイノルズ応力の直応力成分において、渦度が支配的な領域-0.015 < z < 0.002 での改善はみられなかったものの、新しい $k-\epsilon$ モデルを用いると、渦度の主流方向成分 Ω_x 、レイノルズ応力成分 \overline{vw} において全領域で著しい改善がみられ、実験値とほぼ同じ分布が得られた.

また, mod., mod.1の両ケースの比較では両者とも同 程度の改善効果があり, 現段階ではどちらが適当である かの判断ができない. もっとも'Standard' $k - \epsilon =$ デルに対する平均渦度テンソルの影響のみを陽的に与 えるという観点からみれば, $C_{\epsilon 1}$, $C_{\epsilon 2}$ に'Standard' な値を用い, $C_{\overline{q}} = -0.2$ とする mod.1のケースを当 面採用しておいてもよいと考えられる.

5. 結言

乱流モデルにおける消散方程式の再構成の過程と、 その物理的意味についての考察及び係数 C_q の選定を 行い、それを検証するため、従来のモデルである標準 型 $k-\epsilon$ モデル、新しい $k-\epsilon$ モデル mod.および mod.1 による3種類の数値計算とその比較、および実験値と の比較等を行った.

その結果、新しい $k-\epsilon$ モデルを用いると、渦度の主 流方向成分 Ω_x 、レイノルズ応力成分 \overline{vw} において著し い改善が見られ、また実験値ともほぼ一致した.もっ とも、レイノルズ応力の直応力成分での改善はみられ なかった.この点は Boussinesq の表現の修正を含む別 の観点からの改良が必要であろう.

さらに,縦渦を含む船尾の粘性流の推定に応用する ため,新しい k- モデルによる船体周りの数値計算を 行うことも今後の課題としたい.

参考文献

- Davidov, B.I.:On the statistical dynamics of an incompressive fluid, Doklady Academy Nauka SSSR,vol.136,pp.47,1961.
- Hanjalic, K and Launder, B.E.:Contribution towards a Reynolds-stress closure for low-Reynoldsnumber turbulence, J.Fluid Mech, vol. 174, pp. 593-610, 1976.
- 3) Launder, B.E and Spalding, D.B.: The numerical computational of turbulent flows, computer meth-

ods in applied mechanics and enginering,pp.269-289,1974.

- Shih, T.H. and Lumley, J.L.:Kolmogorov behavior of near-wall turbulence and its application in turbulence modelling, Proc.9th Symp, 8-1-1~8-1-6,1993.
- Jones, W.P. and Launder, B.E.: The prediction of laminarization with a two-equation model of turbulence, Int. J. Heat Mass Transfer, vol. 15, pp. 301-314, 1972.
- 6) Chen, H.C. and Patel, V.C.:Near-Wall Turbulence Models for Complex Flows Including Separation, AIAA J,vol.26,pp.641-648,1988.
- Proc.CFD Workshop Tokyo 1994, Tokyo, March 1994.
- Proc.Workshop on Wave Resistance and Viscous Flow, Tokyo, July 1994.
- 9) Tahara, Y.: An Aplication of Two-Layer k- ϵ Model to Ship Flow Computation, J.Society of Naval Architects of Japan, Vol. 177, pp. 161-176, 1995.
- 10) Suzuki, H., Yabushita, K., Toda, Y., Suzuki, T., and Arai, H.: Experimental Study on a Turbulent Boundary Layer with a Longitudinal Vortex - Turbulence Measurement by Triple Sensor Hot Wire -, J. Society of Naval Architects, Japan, No. 225, pp. 47-56, 1996.
- 11) Patel, V.C.,Rodi, W.,and Scheuerer,G.: Turbulence models for near wall and low Reynolds number flows.A review.,AIAA J,No.23,pp.1308-1319,1984.
- Speziale, C.G.:On nonlinear k-l and k-ε models of turbulence.,J.Fluid Mech,No.178,pp.459-475,1987.
- 13) Himeno, Y.:乱流モデルに関する調査,SR 222 資料,1993.
- 14) So, R.M.C Zhang, H.S. and Speziale, C.G.:Near-Wall Modelling of the Dissipation Rate Equation, AIAA J,vol.29,pp.2069-2076,1991.



Fig.3 Comparison of solutions at x=1.0: Contours of the time-averaged velocity \overline{U} .



Fig.4 Comparison of solutions at x=1.0: Contours of the time-averaged velocity \overline{V} .



Fig.5 Comparison of solutions at x=1.0: Contours of the time-averaged velocity \overline{W} .



Fig.6 Comparison of solutions at x=1.0: Contours of the Reynolds stress \overline{uv} .



Fig.7 Comparison of solutions at x=1.0: Contours of the Reynolds stress \overline{vw} .



Fig.8 Comparison of solutions at x=1.0: Contours of the Reynolds stress \overline{uw} .



Fig.9 Comparison of solutions at x=1.0: Contours of the logitudinal vortex Ω_x .



Fig.10 Comparison of solutions at x=1.0: Contours of the turbulent energy k.



Fig.11 Comparison of solutions at x=1.0: Contours of the dissipation rate ϵ .

0.020 Z



Fig.12 Time-averaged velocity \overline{U} distributions at x=1.0.



Fig.14 Time-averaged velocity \overline{W} distributions at x=1.0.



Fig.15 Reynolds stress \overline{uv} distributions at x=1.0.

- Fig.16 Reynolds stress \overline{vw} distributions at x=1.0.
- Fig.17 Reynolds stress $\overline{u}\overline{w}$ distributions at x=1.0.



Fig.18 Longitudinal vortex Ω_x distributions at x=1.0.

Fig.19 Turbulent energy k distributions at x=1.0.

Fig.20 Dissipation rate ϵ distributions at x=1.0.
セールを対象とした粘性流体中における空力弾性問題*1

学生会員西川達雄*2,正会員田原裕介*3,正会員正岡孝治*3,正会員姫野洋司*3

Aeroelastic Behavior of Thin Lifting Membrane Sail

By Tatsuo NISHIKAWA (Student Member), Yusuke TAHARA (Member) Koji MASAOKA (Member) and Yoji HIMENO (Member)

An important factor in the design of lifting membrane is the accurate determination of its aerodynamic behavior. Recent studies ^{2, 4)} related to membrane wings or yacht sails have introduced iterative coupling of two numerical approaches, i.e., vortex-lattice method(VLM) and finite-element method (FEM) for analysis of the flow fields and the membrane deflection, respectively. Although results presented in the works seem to be promising, VLM is not capable for predicting details of flow including viscous effects and three-dimensional-flow separation. This paper presents a numerical approach to predict aeroelastic behavior of lifting membrane in onset viscous flow, where the flow fields are solved by a RaNS equation method. The fully-elliptic RaNS and continuity equations are solved with a zero-equation turbulence model to provide aerodynamic forces acting on the membrane, and FEM considering large deflection theory with membrane-finite elements is used for estimating elastic characteristics of the membrane.

Keywords : Aeroelastic Problem, Membrane, Reynolds-averaged Navier-Stokes Equation, Finite Element Method

1. 緒言

セールはセーリングヨットの推進装置であり、その 性能はセール形状に依存する.セール形状は薄膜とし て空力弾性的に変形し,最終形状はクルーによってさ まざまな方法で制御される.一般的な揚力体において も空力的荷重は揚力体自身の変形やその変形に伴う迎 角の変化に依存し,いわゆる空力弾性的現象が多少な りとも起こっているのであるが,セールの場合は膜状 の揚力面であるためその変形量が大きく,変形量の少 ない他の揚力体とくらべこてこのような空力弾性の影 響が顕著に現れてくる.流体中にあるセールにかかる 流体力の分布はセール形状によって決定されるが,そ の形状は流体力分布などの外力と内部応力の平衡から 定まる.すなわち,このような空力弾性問題を扱うた めにはその両方を同時に知ることが必要であり,この ような干渉問題は解析的に扱うことが困難であり,流 体力分布とセールの変形を同時に数値的に計算するこ とが必要である.

セールを対象とした研究は現在までに多くの研究者に よってなされているが、3次元セールを対象とした初期 の研究として Milgram の研究¹⁾があげられる.この研究 では揚力線理論を用い理想的なセール形状を計算した. 空力弾性を考慮した研究には Jackson²⁾, Atkinson³⁾, Fukasawa⁴⁾等の研究があげられる. Jackson 等²⁾は構 造計算に有限要素法 (FEM) を用い, 流体力には非粘 性流法である渦格子法 (VLM) を用いた空力弾性の計 算手法を確立した. Atkinson 等の研究³⁾ではセールと リグを考慮した研究が行われ、セールシステム全体が 空力に及ぼす影響について考察している.この研究で もほぼ同様の計算方法を用いているが、ここでは四角 形要素を用いている.また,Fukasawa 等の研究⁴⁾でも 計算方法の主要部分はこれらの研究と同じであるが構 造計算に大変形を考慮し増分法を用いることにより動 的な解析もできる計算を行っている.粘性流体を考慮 した3次元セールの研究ではLee 等⁵⁾の研究が挙げら れる.本研究とは直接関係しないが Lee 等⁵⁾は二翼干

^{*1} 平成 10 年 5 月 21 日関西造船協会春季講演会において 講演,原稿受付 平成 10 年 6 月 10 日

^{*2} 大阪府立大学大学院

^{*3} 大阪府立大学工学部

渉問題のシミュレーションを行っている.このような 過去の研究において,粘性流体中の空力弾性問題を解 析した研究は見当たらない.そこで本研究ではセール を対象とした粘性流体中における空力弾性問題を解析 するための計算方法を開発することを目的とした.

セーリングヨットにおいてクルーは与えられたコン ディションのなかで最適な制御を目指すものであるが, 最適な制御をした上ではセール自身の性能が問題とな る. つまりセールデザイナーが考えることは最適な クルーによる制御中のフライングシェープ (帆走中の 形状)を初期形状から予測することである. さらに最 終的にはセール各部に異なった材質を使用する場合, あるいはマスト、ブームやステイ等の境界条件となる セール取り付け部の変位量も考慮にいれた場合といっ た多くの観点を考慮することが必要となってくる、し かし、このように多くの事を考慮しなければならない 場合にも空力弾性的な変形をしているセール単体の正 確な性能を知ることは最も重要な点となり、現在でも 十分に把握できていない点である. セール単体の性能 における風の剥離の問題は注目すべき点である.現在 までの研究で行われていた非粘性流法を用い剥離がな いと仮定する手法には無理があることが分かっている. クローズホールドにおいてもセール表面での風の剥離 がその空力特性に大きく影響していることは Lee 等⁵⁾ の研究にもあるように事実であり、もはや無視するこ とはできないと考えられる.

このような背景のもとに,本研究では剥離を考慮 した粘性流体中の空力弾性の計算手法を開発した. 流体力計算には剥離を表現できる Reynolds-averaged Navier-Stokes(RaNS) 方程式法を用いた.構造計算に は大変形を考慮した有限要素法 (FEM) を用いた.流体 力計算法は著者等が開発したもの⁶⁾に基づいている.そ してこの2つの計算を繰り返し計算にて収束させ定常 解を求める計算手法を開発した.この二つの計算間の データ受渡しには十分な正確さが求められる. さらに, 本研究では上述した実際のセーリングコンディションの 中でクルーによる制御の1つの方法を再現し、その制 御ロープのテンションの方向をパラメータとして揚力, 抗力係数を求めた. さらに迎角をパラメータとした計 算も行い、剥離の影響も考察する.実験との比較など 課題も多いが、セールやハングライダーの設計への応 用の可能性を示し,上述した各パラメータが剥離を含 む空力特性にどのような影響を及ぼすかを試算した.

2. 計算手法

2.1 仮定および計算手法の概要

本研究では,任意のセールに対してある制御パラ メータの最適値を知ることを目的としたので,リグを 含むセールシステム全体における設計パラメータは省 いた.以下にその詳細を記す.まず第一に,リグの影 響は考慮はせず,マスト,ブームは剛と仮定した.バッ テン等のセール自体の付加物も考慮せず,セールは純 粋な薄膜状の揚力面とした.また,素材は等方性とし, 変形は弾性変形とした.つぎに,セールに働く流体力分 布は以下に示す仮定のもとで計算を行った.セール周 りの流体は非圧縮とし,粘性を考慮した RaNS 方程式 法を用い計算を行った.セールの厚さの影響,流体中 のマスト,ブームは考慮しない。本研究では流体力計 算と構造計算を繰り返し計算にて収束させ平衡状態を 求める計算手法を用いた.物理空間における座標系と して Fig.1に示すように船体固定座標を用い,ブームは 常に船体方向つまり xz平面と平行にあるものとした.

2.2 流体力と構造変形のカップリング計算

本研究では流体力計算と構造計算を交互に行い,反 復計算により平衡状態を求めた.形状および流体力分 布の情報をその二つの local computation 間で十分な 正確さをもって受渡しする必要がある.流体力,構造 計算のカップリングの手法をフローチャート Fig.2に示 す.まず,ある初期形状から流体力計算用の格子を生 成し,流体力分布を計算する.その流体力分布と制御 による外力を構造計算用の要素に入力として与え,そ の外力と内力が平衡状態となる形状を計算する.そし て変形した形状から再び流体力計算用の格子を生成す る.このような繰り返し計算を変形量,流体力分布が 十分に収束した状態になるまで行う.

流体力計算,変形計算の local computation 間での 変数のデータ受渡しは cubic spline により補間を行っ た.その際に出て来る誤差は十分に少なくなければな らない.実際の計算においては構造計算用節点から流



Fig. 1 Global coordinate system and the definition of the direction of the sheet tension



Fig. 2 Flow chart of whole calculation

体力計算用格子への形状データの受渡しは補間するこ とにより行い流体計算用格子から構造計算用節点への 形状情報の受渡しは行わず,前のデータを使用するが, 補間の際に出る誤差の目安として,構造計算用節点→ 流体力計算用格子→構造計算用節点と補間を行い元の 構造計算用節点との差の各節点での合計

$$\|d\mathbf{U}\| = \sum_{i \in N_i} (dUx_i^2 + dUy_i^2 + dUz_i^2)^{1/2}$$

を計算した。このノルム $||d\mathbf{U}||$ は全ての補間過程にて 無次元長さで 1.0×10^{-5} 以内に抑えた.ここで $N_i \equiv$ セール表面上の全節点とする.

2.3 流体力計算

本研究で用いた粘性流場計算法は著者らの一人が 開発したもの⁶⁾に基づいている.3次元非定常,非 圧縮流体に関する無次元化された Reynolds-averaged Navier-Stokes(RaNS) 方程式は,デカルト座標を用いて 無次元化されたテンソル表記により以下のようになる.

$$\frac{\partial U_i}{\partial x^i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \left(U_j \frac{\partial U_i}{\partial x^j} + \frac{\partial \overline{u_i u_j}}{\partial x^j} \right) + \frac{\partial \hat{p}}{\partial x^i} - \frac{1}{Rn} \nabla^2 U_i = 0 \quad (2)$$

$$\nabla^2 = \frac{\partial^2}{\partial x^j \partial x^j} \tag{3}$$

ここで $U_i = (U, V, W)$ および $u_i = (u, v, w)$ は代表速度 U_0 で無次元化された平均および変動速度成分,

 $x_i = (X, Y, Z)$ は代表長 Lで無次元化された座標, $\hat{p} = (p + \rho g Y) / \rho U_0^2 \iota \rho U_0^2$ で無次元化された piezometric pressure, $Rn = U_0 L / \nu \iota \nu \wedge J \nu \chi$ 数, $-\overline{u_i u_j} \iota U_0^2$ で無次元化されたレイノルズ応力成分である.なお, 代表長 L は今後すべてにおいて最大コード長とした. 3 次元剥離を伴う解析に問題は残されているもののこ こでは乱流モデルに簡便な Baldwin-Lomax 乱流モデ ルを用いた.

輸送方程式の離散化には有限解析法による12点離散 化方程式を用い,速度場-圧力場結合にはPISOタイ プ/1ステップ法を採用した.基本方程式はすべて非 定常で定義されているが,本研究では定常解を求める ことを目的とし時間(t)を収束パラメタとして用いた.

流体力計算用の格子は各ステップ毎に更新し格子数 はセールの長さ方向に120, 鉛直方向に30, 向射方向 に40取り, 144000とした.全体像はFig.3に示す.

また流体力を求めるに当たっては剥離を伴う計算を 行っているので流体力分布は振動している. そこで本 研究の目的は平衡状態を求めることにあるので十分に 定常振動に至った流体力分布の一周期分の平均をとり,



Fig. 3 Grid system for fluid calculation



Fig. 4 One period of the vibration of the fluid calculation



Fig. 5 Comparison of membrane deformation between Hangai's result and present method



Fig. 6 Arrangement for structural analysis calculation

それをそのステップでの流体力分布とした.その一例 として圧力揚力係数を Fig.4に示す.縦軸は圧力揚力 係数で横軸は無次元時間 t である.なお,各計算で一 周期分のデータ数は約60個である.

2.4 構造計算

本研究では構造計算に通常の有限要素法を使用し要素は三角形薄膜要素を使い大変形を考慮した. 増分法 により解析し, Newton-Raphson法によって収束計算 を行った. また安定化のため曲げ剛性を考慮した.

本研究の構造計算の手法が薄膜に対して有効に使用 できるか否かの妥当性は、半谷等による一定圧力下 の正方形膜の中央断面の弾性変形の計算結果⁸⁾と比較 することにより判断した.計算状態は一辺18mの正 方形, 膜材の剛性は $E = 9.0 \times 10^3 kgf/mm^2$, 膜厚は 1.0mm, ポアソン比は 0.45, 内圧 $100kgf/m^2$ である. Fig. 5 に示したように,本研究で使用した 3 次元膜大 変形計算は半谷らによる計算と良い一致を示しており 本研究で用いた構造計算の手法の妥当性を確認した.

以下のケーススタディーで用いたセール構造解析用 の計算要素数は 899 個, 節点数 1680 点である. Fig. 6 にその全体像を示す.

3. 計算結果

3.1 ケーススタディーの方法

本研究ではケーススタディーを以下の条件のもと 行った.セールのリーディングエッジはすべて固定し, セール下部は自由にし,制御用テンションを一点にか けた.制御用テンション Tは 1000kgfで一定とし,方 向のみを変えた.テンションの方向は Fig. 1で示すよ うに定義し, xz平面に平行とし, x 軸との角度をθとし た.これは,実際のヨットではジブセールならばジブ リーダーの位置に相当し,メインセールならばジブ リーダーの位置に相当し,メインセールならばアウト ホールとメインシートにかかるテンションの合力の方 向に相当する.詳しいセールの初期状態は Table 1 の とおりであり,風 (Table 2) は一様流とした.なお,こ れらの条件は Fukasawa 等が行った計算結果と比較す るため同一のものとした.

3.2 収束状況

流体力,構造計算の繰り返し計算の過程において流体力計算,構造計算を各1回づつ終ると全体の計算で 1ステップ終了とした.収束状況の例をFig. 7,8に示

Table 1 Principal dimension of the sail

Sail	Length	11.70~(m)
	Width	$4.100 \ (m)$
	Elastic Modulus	$630.0 \; (kgf/mm^2)$
	Poisson's Ratio	0.30
	Density	$1140.0 \ (kg/m^3)$
	Thickness	$0.50 \ (mm)$
Mast	Material	Rigid

Table 2 Wind condition

Wind	Velocity	10.0(m/s)
	Attack Angle	15.0(deg)
	Air Temperature	20.0(c.deg)
	Air Density	$1.205(kg/m^3)$
	Coeff. of Viscosity	$1.82 \times 10^{-5} (Pa \cdot s)$
	Eddy Viscosity	$1.50 \times 10^{-5} (m^2/s)$
	Reynolds Number	$2.726546 imes 10^{6}$

す. 横軸はステップ数n、縦軸は代表長Lで無次元化 された変形量の各節点での合計 $||U^n||$ 、揚力係数 C_L で ある. なお、変形量の定義は以下の様にした.

$$||U^{n}|| = \sum_{i \in N_{i}} \{ (x_{i}^{n} - x_{i}^{1})^{2} + (y_{i}^{n} - y_{i}^{1})^{2} + (z_{i}^{n} - z_{i}^{1})^{2} \}^{1/2}$$
(4)

また,以下に揚力係数CL,抗力係数CDの定義をする.

$$C_L = \frac{Lift}{\frac{1}{2}\rho U_\infty^2 S} \tag{5}$$

$$C_D = \frac{Drag}{\frac{1}{2}\rho U_\infty^2 S} \tag{6}$$

ここで Sはセール面積とする. 平衡状態に達したかどう かの判定は,変形量,流体力の各係数等から判断され, Fig. 7,8のようにそれぞれ約5回で変動がなくなった のでここで計算を打ち切りn = 5を平衡状態とした.

3.3 計算結果と考察

まず、変形量に関しては Fig.9に初期形状からの y方向の変位量を示す. 左から順に $\theta = 35^{\circ}, 45^{\circ}, 55^{\circ}$ となっている. テンション角度が少ない方がよりトレーリングエッジが開いていることがわかる. これは実際のセールにおいてジブセールならジブリーダーを後ろにさげたことに対応しており実現象と良く合致している.また、Fig.10は最大主応力分布図で同じく左から順に



Fig. 7 An example of step history of the deflection of the global iteration



Fig. 8 An example of step history of the lift coefficient of the global iteration

 $\theta = 35^{\circ}, 45^{\circ}, 55^{\circ}$ となっている.これはテンション角度 が少ない程セール下部にストレスが入っており,テン ション角度が大きい程リーチにストレスが入っている ことが良く現れ,定性的に正しいと思われる結果が出 ている.

次に Fig.11,12,13は縦軸にそれぞれ揚力係数 C_L ,抗 力係数 C_D ,揚力抗力比 $L/D \varepsilon$,横軸にテンション角 度 θ を示した. θ =35°で最大揚力係数,最小抗力係数, 最大揚力抗力比をとっており,このセールでのこの制御 方法での最大効率は制御パラメータ θ =35°で得られる ことが分かった.また,非粘性流法を用いた Fukasawa 等の計算結果と比較すると揚力係数ではかなり低く, 抗力係数ではかなり高い結果が得られている.この原 因としてまず考えられるのは剥離の影響であろうと思 われる.

4. 剥離の影響について

前章での議論の中で非粘性流法を用いた Fukasawa 等の計算結果とかなり違う計算結果を得た原因として 剥離の影響だと考えたが,本手法での計算上のセー ル表面の流線 Fig.15を見ると,これはテンション方向 $\theta = 45^\circ$ の時のサクションサイドの表面流線だが確かに



Fig. 9 Distributions of the deformed value of yaxis for three cases ($\theta = 35^{\circ}, 45^{\circ}, 55^{\circ}$)



Fig. 10 Distributions of the principle stress for three cases ($\theta = 35^{\circ}, 45^{\circ}, 55^{\circ}$)



ほぼセール全面で剥離が見られる. そこで同じ形状で 迎角 (α) を変えた時の流場を見ると $\alpha = 8^{\circ}$ (Fig.16) の ときは $\alpha = 15^{\circ}$ にくらべかなり剥離が押えられている ことがわかる. $\alpha = 0^{\circ}$ (Fig.17)の時にはほとんど剥離 がなくなり、裏面で剥離が確認できる. Fig.14はこの ときの揚力係数 C_Lを示したものであるが,縦軸,揚力 係数CL, 横軸, 迎角とし Fukasawa 等の計算結果と比 較してある. 揚力傾斜係数 C_L/α で見てみると $\alpha = 8^\circ$ の時には Fukasawa 等の計算結果に近い物が得られて いることが分かる. つまり, Fukasawa 等の計算では通 常考えられている迎角を使用していて剥離の影響を無 視できるという仮定を入れており本計算手法でも剥離 の少ないときには非粘性流法を用いた Fukasawa 等の 計算結果に近い揚力係数が得られるものの, この初期 条件においてはα = 15°のときには剥離の影響を無視 できないと結論づけることができる.しかし,実際の セーリングコンディションにおいて本計算手法で行っ



Fig. 14 Lift coefficients for different attack angles (Comparing with Fukasawa's result)

た計算結果のように $\alpha = 15^{\circ}$ でサクションサイドでほ ぼ全面剥離が見られるかどうかは疑問である.この理 由として本計算手法の妥当性の問題以前に本研究で用 いた初期セール形状,制御方法,乱流モデルの問題な どがあると思われこれらは今後の課題とする.

5. 結言

本研究では流体力計算には RaNS 方程式法を用い, 構造計算には大たわみを考慮した FEM を用いた空力 弾性問題の数値計算を行い以下の結論を得た.

- 1. 剥離を考慮した粘性流体中のセールを対象とした 空力弾性の計算手法を開発した.
- ケーススタディーを行いセールの最大効率をある 制御パラメータで得られることを確認した.
- 3. 非粘性流法を用いた Fukasawa 等の計算結果と比 較することにより、剥離の空力特性に及ぼす影響 を明らかにした.

これらの結論より、シミュレーションの条件を考え 実験などの実現象との合致を確認するなど、多くの課 題は残されているものの、本研究で用いた計算手法が セール設計への有効なツールとなりうることを示した.

参考文献

- Milgram, J. H.: The Analytical Design of Yacht Sails, The Society of Naval Architects and Marine Engineers, 1967, pp118-160.
- Jackson, P.S. and Christie, G.W.: Numerical Analysis of Three-Dimensional Elastic Membrane Wings, AIAA Journal, Vol.25, No.5, 1987, pp676-682.
- 3) Atkinson, P. and Szantyr, J. A.: An integrated approach to the analysis of mast-sail systems, The



Fig. 15 _Stream lines and the pressure counters of the surface of suction side ($\alpha = 15^{\circ}$)



Fig. 16 Stream lines and the pressure counters of the surface of suction side ($\alpha = 8^{\circ}$)



Fig. 17 Stream lines and the pressure counters of the surface of suction side ($\alpha = 0^{\circ}$)

-249-

Royal Institution of Naval Architects, 1991, pp73-89.

- Fukasawa, T. and Katori, M.: Numerical Approach to Aeroelastic Responses of Three-Dimensional Flexible Sails, Proc. The 11th Chesapeake Sailing Yacht Symposium, Chesapeake, January, 1993, pp87-105.
- Lee, Y., Miyata H. and Sato T.: CFD Simulation of Two-Sail Interaction about Sailing Yacht, J. Society of Naval Architects of Japan, Vol.181, 1997, pp25-44.
- Tahara, Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, J.Kansai Society of Naval Architects, Japan, No.226, September 1996, pp63-76.
- O.C.Zienkiewicz, and R.L.Taylor : The Finite Element Method 4th Edition, Vol 1. Vol 2., McGRAW-HILL Book Company 1989.
- 8) 半谷裕彦, 鈴木俊雄, 関富玲: 変位制限をもつ膜構造の解析,構造工学における数値解析シンポジウム論文集,第13巻, 1986, pp83-88.

討 論

[討論] (金沢工業大学) 深沢 塔一

私の計算でもそうであったが計算で得られた主応力分 布や変形は必ずしも実際のセールと合っていない.(お そらくそのために迎角15°でも剥離を生じている.)こ の原因の1つにFEM要素に三角形要素を用いている ことがあるのではないかと考える.この点については 検討された方がよいと思う.また,リーチやラフの張 力の入れ方も影響するのでこれについても確認してみ たら如何か?

[回答]

変計量については確かに初期形状から y方向にへこん でいる部分が多く実際のセールとは合っていないとこ ろがある.つまり,セールがより平板に近づき,リー ディングエッジに対する流入角が大きくなっている. そのため迎角15°でも剥離を生じているのは私達も同 じ意見である.もう一つの原因として今回は深沢先生 等の計算結果と比較するため同じ形状を使用したが, 実際のセールに流体力を合わせるためにはより優れ たセール形状で計算し御指摘の点,制御方法,シミュ レーションの初期条件などを検討し,その上で FEM 要素などについても検討する必要があると思う.

RaNS法によるアメリカ杯レース艇のストラット/バルブ近傍流場の 数値計算および実験との比較*1

正会員田原裕介*2,学生会員吉田豊*3,学生会員西田隆司*3

An Application of RaNS Equation Method to Strut/Bulb Configuration of America's Cup Sailing Yacht and Comparison with Experiments

By Yusuke TAHARA (Member), Yutaka YOSHIDA (Student member) and Ryuji NISHIDA (Student member)

A numerical method is developed for calculating boundary-layer and wake flows around a strutkeel/bulb system of the International America's Cup Class (IACC) sailing yacht. The Reynoldsaveraged Navier-Stokes and continuity equations are solved with the Baldwin-Lomax turbulence model, using a body conforming grid, finite-analytic discretization, and a PISO-type velocity-pressure coupling algorithm. In addition, circulating water channel experiments are performed for hydrodynamic force measurements and flow visualization on the body surface. An overview is given for the present numerical approach, and numerical results are presented and discussed for two-types of test strut-keel and bulb models, including detailed comparison with the experimental data.

Keywords : America's Cup Sailing Yacht, Reynolds-averaged Navier-Stokes Equations, Strut-Keel/Bulb System, Hydrodynamic Forces, Flow Visualization

1. 緒言

アメリカ杯レース艇に代表される高速帆走艇は,一 般のヨットに比べ極めて大きい面積のセールを有し,そ のヒーリングモーメントと釣合わせるために大重量の バラストバルブを必要とする.バルブの設計要件は低 重心,モーメントバランスなど数多くあるが,最も基 本的な要素は極力小さな抵抗特性を有することである. Hullの場合と異なり,実船スケールにおける乱流遷移点 の推定が困難であるなどの問題があり,模型試験によっ て実船搭載時の性能を正確に予測することが難しい.

現在の IACC(International America's Cup Class) ルールが適用された 1992 年アメリカ杯レース以降, IACC ヨット用バラストバルブは参加各シンジゲート によって検討されてきた.形状もかなり研究され,バ ルブ単体の直進時の抵抗成分のほとんどは摩擦抵抗が 占めるレベルであると考えられる.しかしながら通常 ヨットは Leeway 角をもって帆走するため,流入迎角 がある場合において,またキールストラット (以下スト ラットと略称)と一体化した状態で低抵抗特性を保持 するバルブ形状の開発が重要とされている¹⁾.

近年の計算流体力学 (CFD) 技術の向上に伴い, 模型試験コストの削減および流体力の高精度予測を目的 とした, IACC ストラット/バルブ用 CFD 設計ツール の開発が強く望まれている^{2),3)}. この CFD ツールは 物体表面の3次元剥離流場や,ストラットとの接合部 流場を詳細に計算する必要があり,そのためにはレイ ノルズ平均ナビエ・ストークス方程式法 (RaNS法)の 適用が望ましい.ストラット/バルブ形状を対象とし た CFD 手法, また実験的研究は幾つか報告されてい る⁴⁾⁻⁹⁾.しかしながら, RaNS 法に基づき,またスト ラット/バルブ近傍の境界層流場の高精度予測を目的と した手法,さらにその解の実験値との詳細な比較は, 現在までに報告されていない.

本研究では,著者らが開発を続ける RaNS 法スキー ム¹⁰⁾⁻¹²⁾を拡張し,IACC ヨット用ストラット/バルブ 設計支援ツールを構築した.さらに回流水槽における模 型実験を行い,物体近傍流場および流体力の計算予測

^{*1} 平成10年5月21日 関西造船協会春季講演会におい て講演,原稿受付、平成10年6月10日

^{*2} 大阪府立大学工学部。

^{*3} 大阪府立大学大学院工学研究科



Fig. 1 Definition sketch of coordinate system



Fig. 2 Computational domain and boundaries

精度を評価した. この CFD ツールは CAD(Computer Aided Design) ツール, さらに流場情報を抽出するポ ストプロセッサーと結合しており, 任意のストラット/ バルブ形状に対応できる.以下,本 CFD ツールの概 要を述べ,2種のテストモデルに関する結果を用い, 本 CFD ツールの有効性を検討する.

2. 計算手法の概要

ここでは、本研究で用いた計算手法の概要を述べる. 基礎方程式はデカルト座標系 (Fig.1) で記述された、3 次元非定常、非圧縮流体に関する RaNS 方程式である. 乱流モデルには Baldwin-Lomax(BL) モデル¹³⁾を用い ている.数値計算においては、基礎方程式を数値的に生 成した非直交曲線座標系に変換し、さらに輸送方程式を 有限解析法により離散化する.速度場/圧力場結合には PISO タイプ/1 ステップ法を採用する.以上の方法に より、(X,Y,Z) 軸方向の平均流速場 (U,V,W), 圧力場 p,および渦動粘性係数 ν_t が各計算格子点で求まる.数 値計算法の詳細は著者らの文献¹⁰⁾⁻¹²を参照されたい.

本研究で用いた BL モデルは、物体近傍流場の解析 および流体力評価に関しては妥当な精度を有すること が報告されている¹⁴). BL モデルの Length Scale は物



Fig. 3 Overview of computational grid

体表面からの距離に基づいており,接合部近傍では特別な配慮が必要になる.本研究では,ストラット/バルブ接合部近傍でレtを評価する際,双方の表面からの距離に基づくレtを求め,その距離を用いた加重平均によって目的地点のレtを決定している.

次に境界条件および初期条件について述べる.計算領域を Fig.2 に示す. Fig.2 の境界面 Index に従い,本計算法で用 いた境界条件を記述すると次のようになる:On So(外境界), $(U, V, W, \partial p/\partial \eta) = (U_0 \cos\alpha, U_0 \sin\alpha, 0, 0);$ On Sb($\checkmark \lor \lor \lor$ ブ表面), $(U, V, W, \partial p / \partial \eta) = (0, 0, 0, 0);$ On Ss(スト ラット表面), $(U, V, W, \partial p/\partial \zeta) = (0, 0, 0, 0)$; On Se(流 出境界), $(\partial U/\partial \xi, \partial V/\partial \xi, \partial W/\partial \xi, \partial p/\partial \xi) = (0, 0, 0, 0);$ On St(上部境界), $(\partial U/\partial \eta, \partial V/\partial \eta, \partial W/\partial \eta, \partial p/\partial \eta) =$ (0,0,0,0). ここで $(U_0 cos \alpha, U_0 sin \alpha, 0)$ は一様流ベクト 向,放射方向,ガース方向(周方向)にとった計算座標 系である.計算座標系は計算格子の配列方向と同一で あることに注意されたい. なおバルブ表面における η 軸およびストラット表面における(軸は、それぞれの 表面の法線方向と一致するように格子生成を行ってい る.一方,初期条件は流場全域で一様流条件、すなわ ち $(U, V, W, p) = (U_0 cos \alpha, U_0 sin \alpha, 0, 0)$ を用いている.

3. 計算および実験結果

3.1 供試模型および計算/実験条件

本研究で用いた供試模型は, IACC ヨットに実装され たストラット/バルブ形状を参考にし,研究目的のため に形状変更し作成した2種のテストモデル (Bulb-types A and B)である.なおストラット部は同一形状であ る.A·B型の全長は同一(0.66m)とし,B型はA型よ り体積および表面積ともに小さく,全体的により細長 型である.また,今回使用したバルブ形状は全て上下



Fig. 4 Comparison of frictional streamlines between computation and experimental observation (Bulb-type A, $\alpha=0^{\circ}$)

対称である.

計算に用いた格子数は (ξ, η, ζ) 方向にそれぞれ (160,40,56)である.これは計算精度,今回の研究で利 用可能な計算環境,設計サイクルでの実用的計算効率 を考慮し,最適化された値である.計算格子の一例を Fig.3 に示す.また今回考慮した全ての計算条件におい て、5 無次元時間までに解は収束した.5 無次元時間に 要した計算時間は,DEC-AlphaStation300を用いて約 24 時間である.一方,IACC ヨットのストラットには タブと呼ばれる一種のフラップ機構が装備されており, 本計算法では Fig.1 に示すようにタブ角 (β)を考慮し た計算も可能である.しかしながら,本論文では $\beta=0^\circ$ の結果のみを検討する.

表面近傍流場の可視化実験および流体力試験は,大阪府立大学回流水槽で行った.実験におけるバルブ没水位置は水面下約50cmである.本研究では2種のバルブの相対評価に焦点を当てたため,水面板設置などの造波影響除去は行っていない.実験における平均レイノルズ数(代表長はバルブ全長)は,流場可視化実験および流体力試験においてそれぞれ Rn=650,000 および700,000 であり,計算においても同条件とした.またバルブ先端近傍およびストラット前縁近傍で乱流促進を施している.一方,計算では流場全域を乱流場と仮定した.

Fig. 5 Comparison of frictional streamlines between computation and experimental observation (Bulb-type A, pressure side, $\alpha=4^{\circ}$)

3.2 表面流場の比較

ここでは計算で求めた Frictional streamline および実 験で観測した表面流場を比較し,物体近傍流場を調べる. なお実験における可視化手法には油膜法を用いている. 油膜法で観測された流場パターンは,厳密には粘性底層 より外側の流場の影響を受け,計算で求める Frictional streamline と比較した場合,結果がやや異なることが 予想される.しかしながら,表面近傍流場の挙動を大ま かに把握する観点では,両者は比較可能であると考え られる.以下,計算値および実験値ともに限界流線で あるとして議論する.Figs.4~6は Bulb-type Bに関す る,それぞれ迎角 α =0° Side view, α =4° Pressure-side view, そして α =4° Suction-side view の実験値と計算 値の比較である.また Fig.7 には α =4° Suction side か ら見たストラット後縁近傍/バルブ上部の限界流線を示 す.以下でそれらの結果を考察する.

まず迎角α=0°の結果を検討する (Fig.4). 実験で観 測されたストラットおよびバルブ上の限界流線は,ス トラット自由表面近傍を除き,計算結果とほぼ一致し ている.バルブ上の流線は物体先端部から後端部まで 連続し,回転楕円体の場合¹⁵⁾のような物体後端部近傍 の3次元剥離は見られない.バルブ形状が上下対称で あるにも関わらず,流線パターンには上半部/下半部で 若干の相違が見られ,これはストラットの影響である と考えられる.バルブ単体の厳密な性能評価には,迎



Fig. 6 Comparison of frictional streamlines between computation and experimental observation (Bulb-type A, suction side, $\alpha=4^{\circ}$)

角がない場合においても、ストラット影響を考慮する 必要があることを示唆している.

次に迎角 $\alpha=4^{\circ}$ の結果を考察する (Figs.5,6). 迎角 α=0°の場合と同様,実験で観測した限界流線は,自由 表面近傍を除いた全領域で計算結果とほぼ一致してい る. また迎角が無い場合は対称であった Pressure side および Suction side(以下 P 側および S 側と略称)の限 界流線が、迎角α=4°場合は P 側と S 側で大きく異なる ことが示されている.ストラット前縁以降のバルブ表面 の、特にバルブ中/下部領域において、限界流線は P 側 では下方向へ, また S 側では上方向へ向かう傾向が見 られる.これは上下対称なバルブ単体形状では起こり 得ず,明らかにストラットの影響である.この現象は, Fig.8に示すような翼端渦伝播モデルによって大まかに 理解することができる. すなわち, ストラットの翼端渦 は翼端のP側からS側へ向かう流れを誘起し、それが バルブ表面流場に影響を与え,上述した限界流線の傾 向を示すに至ったものと考えられる.計算値の流場を 用い,バルブ前方より計算した流線の概観を Fig.9 に 示す.バルブ後方に伝播する縦渦の存在が確認できる.

最後にストラット/バルブ接合部近傍の流場を考察 する (Fig.7). ここでは特にストラット後縁近傍の流場 に着目する.上述した物体側面における結果と同様, 実験で観測された流場パターンは計算結果とほぼ一致 している.実験結果では,ストラット後縁/接合部近傍



Fig. 7 Comparison of frictional streamlines between computation and experimental observation (Bulb-type A, near bulb tail, $\alpha=4^{\circ}$)

に油膜が停滞する部分があり、これは Davenport¹⁶)ら も実験で示した'Small Separated Zone'であると考 えられる.計算結果ではその領域に 3 次元剥離領域が 存在し、Focal 型の特異点も見られる.ストラット後縁 以降、実験値では翼/平板接合部流場で典型的に観測さ れる逆 V 字型の流線パターン (例えば Davenport¹⁶) が認められるが、その様子は計算結果でもほぼ正確 に再現されている.実験値において、この逆 V 字型 パターンは迎角 α =0°の場合は左右対称であるが、迎 角 α =4°の場合は P 側から S 側へややずれた様子が見 られ、同様な傾向が計算結果においても確認できる (Fig.7 中 (A) および (B)).

3.3 流体力の比較

ここでは、流体力実験および計算の結果を考察する.対象としたのは前述した Bulb-type A および B である.(株) ニッポンチャレンジ技術開発委員会の Non-Disclosure Agreement に従い、以下の議論における実験値および計算値の流体力係数は、一定の揚力係数 CL_M および抗力係数 CD_M で除した値、すなわち CL^* および CD^* で表示する.

まず迎角を変化させた場合の抗力係数の比較を Fig.10 に示す.ここで考慮した迎角範囲において,各迎角に おける2種のバルブ間の抵抗順位は,実験値と計算値



Fig. 8 Sketch of circulation effects around a strut/bulb configuration for nonzero attack angle



Fig. 9 Computed streamlines around Bulb-type B (α =4°, Pressure-side view)

で一致している.実験値における A 型の抗力増加率は B 型より小さく,また同様な傾向が計算でも予測され ている.一方,計算値と実験値の定量的な差(例えば $\alpha=0^{\circ}$ において平均約5%)は,実験における乱流促進 の方法,また自由表面影響などが考えられる.しかし ながら,2種の供試模型のストラット形状は同一で, かつ前述した表面流場観察の結果では自由表面影響が 水面近傍のみで顕著であることから,実験で示されて いる A-B 間の相違は主にバルブの粘性抵抗の差である と考えられる.したがって本手法は,バルブ形状の変 化による抵抗値順位の推定という目的に関しては,妥 当な精度を有していると考えられる.

次に,迎角を変化させた場合の揚力係数の比較を行う(Fig.11).上述した抗力係数の場合と同様,今回考 慮した迎角範囲において,実験値で得られた各迎角に おける2種のバルブ間の揚力順位は計算結果と一致し ている.実験値および計算値共にB型の揚力増加率は A型のものより大きい.誘導抵抗の増加が全抵抗成分 の増加に関与することを考慮すれば,揚力増加率が大 きいB型の抵抗増加率がより大きいことになり,前述



Fig. 10 Comparison of drag coefficients between computations and measurements (Bulbtypes A and B)



Fig. 11 Comparison of lift coefficients between computations and measurements (Bulbtypes A and B)

した抵抗値の傾向と合致する結果である.実験値と計 算値の間の定量的な差に関しても,上述した理由など が考えられる.

4. 結言

本研究では, RaNS 法に基づく IACC ヨット用スト ラット/バルブ設計支援ツールを構築した.さらに回流 水槽における模型実験を行い,計算値を実験値と比較 することによって,物体近傍流場および流体力の予測 精度を評価した.その結果,本手法は実験で観測され た物体近傍流場の重要な挙動を正確に予測でき,さら に流体力評価に関しても,バルブ形状変化に伴う流体 力順位,および流体力増加傾向を正確に予測できるこ とが確認された.

最後に、本研究を行うにあたり、種々のご助言ご協 力をいただきました大阪府立大学姫野洋司教授、神戸 商船大学戸田保幸助教授、(株)三井造船昭島研究所松 井亨介氏、(有)SEALS上野一郎氏に感謝の意を表わし ます. なお本研究の一部は(株)ニッポンチャレンジ開 発技術委員会/アペンデージチームとの共同研究として 行われました. 委員各位, さらに神戸商船大学学生各 位の御助言に謝意を表します.

参考文献

- Larsson, L. and Eliasson, R.E.: Principles of Yacht Design, A and C Black Publishers Ltd., London, 1994
- 広嶋文哉,宮田秀明,秋元博路: CFD 航走シミュ レーションによる帆走艇の設計法,日本造船学会 論文集,第181号,1997, pp. 15-23
- 片岡秀太郎,宮田秀明,金井亮浩: CFD 航走シミュレーションによる航走艇の設計法 (第2報),日本造船学会論文集,第182号,1997, pp. 113-120
- 4) Rosen, B.S., Laiosa, J.P., Davis, W.H. and Stavetski, D.: SPLASH Free-Surface Flow Code Methodology for Hydrodynamic Design and Analysis of IACC Yachts, Proc. The Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, 1993, pp. 35-49
- 5) Rosen, B.S.: SPLASH Nonlinear and Unsteady Free-Surface Analysis Code for Grand Prix Yatch Design, Proc. The Thirteenth Chesapeake Sailing Yacht Symposiumm, Annapolis, 1997, pp. 211-225
- 6) Farmer, J., Martinelli, L. and Jameson, A.: A Fully Viscous Nonlinear Free-Surface Analysis Tool for IACC Yacht Design, Proc. The Twelfth Chesapeake Sailing Yacht Symposium, Annapolis, 1995, pp. 157-170
- 7) 平田信行,日野孝則,不破健:水中翼とストラット が付いた没水体まわりの流れ場について,関西造 船協会誌,第217号,1992,pp.45-49
- 8) 増田聖始,笠原良和,芦立勲:境界要素法による翼付き没水体まわりの流れ計算,日本造船学会論文集,第179号,1996, pp. 1-9
- 9) 今井新,内山幸樹: IACC 帆走艇のキールシステム に関する研究,卒業論文,東京大学工学部船舶海洋 工学科, 1994
- 10) Tahara, Y.: Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments, 関西造船協会誌, 第 220 号, 1993, pp. 29-47
- Tahara, Y.: Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht, 関西造 船協会誌, 第224号, 1995, pp. 1-11

- 12) Tahara, Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, 関西造船協会誌, 第 226 号, 1996, pp. 63-76
- Baldwin, B.S. and Lomax, H.: Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, 1978, pp. 1-8
- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994
- 15) Tahara, Y., Mitarai, S. and Himeno, Y.: A Computational Study of Three-Dimensional Laminar Separation on a Propate Spheroid at Incidence, 関西造船協会誌, 第 225 号, 1996, pp. 93-105
- 16) Davenport, W.J. and Simpson, R.L.: Some Time-Dependent Features of Turbulent Appendage-Body Juncture Flows, Proc. Sixteenth Symposium on Naval Hydrodynamics, Berkeley, 1986, pp. 312-335

論

[討論] (運輸省船舶技術研究所)日野 孝則

討

1. 流体力の比較における実験と計算のレイノルズ 数および乱流遷移の扱いについて教えてください.

2. 揚力の計算値は計算領域の大きさに依存すると 思いますが,外部境界はどこに設定されているので しょうか.

[回答]

1. 設定レイノルズ数は本文に記述したとおりです. また乱流遷移の扱いについても同様です.

2. X 軸まわり半径1代表長の位置に円筒型外部境 界を設定しています.

[討論] (住友重機械工業(株)) 佐々木 紀幸

計算は自由表面を剛体に置き換えて,回流水槽の自 由表面付の実験と比較されていますが,鏡像と見なし て良いかどうかを判断するために,例えば計算上のス トラット長さを変更するなどしてその影響を考察して おられたらご教示ください.

[回答]

本研究で目的としたのは,異なる形状を有するバル ブの相対評価です.よって実験値に含まれる自由表面 影響の検討は行っていません.対象とした2種のモデ ルのストラット形状は同一であり,造波においてスト ラットの影響が支配的であるとすれば,2種のモデル 間の流体力の差は,主にバルブ形状の差によるもので あると考えられます.バルブの没水深度および限界流 線の状態より,自由表面影響はバルブ性能の相対関係 を逆転させるほど大きくないと考えています.

CFDによるタンカー船型の船尾形状最適化 - 第1報:粘性抵抗最小化 - *1

正会員田原裕介*2,正会員齋藤泰夫*3,正会員姫野洋司*2

CFD-Aided Optimization of Tanker Stern Form - 1st Report: Minimization of Viscous Resistance -

By Yusuke TAHARA (Member), Yasuo SAITOH (Member) and Yoji HIMENO (Member)

This paper presents optimization of tanker stern form based on viscous flow information obtained from Computational Fluid Dynamics. The numerical method is based on coupling of Reynoldsaveraged Navier-Stokes equation solver and nonlinear programming technique, i.e., finite-analytic discretization and PISO-type velocity/pressure coupling method, and successive quadratic programming, respectively. In this 1st report, minimization of viscous resistance is considered and stern form of a given tanker hull is optimized with inequality constrains such that displacement of the modified hull is equal or larger than that of the original, and the profile and maximum beam length as well as depth of the modified hull are same as those of the original. In the following, overview of the present numerical method is described and results are presented for optimization of SR221b tanker hull, including discussion of identification of salient differences of geometries and flows between the original and optimized hull forms. In addition, trends of modification of the stern form are evaluated by a viewpoint of tanker-hull-form designer. In conclusion, the present method appeared to successfully optimize the given tanker stern form, and a good agreement is demonstrated between trends displayed in the present modification and those commonly in use in actual tanker hull form design.

Keywords : Ship Hull Optimization, Tanker Hull Form, Reynolds-averaged Navier-Stokes Equations, Nonlinear Programming, Successive Quadratic Programming, Viscous Resistance

1. 緒言

タンカー船型の設計において,船尾形状の決定は最 も重要な作業の一つである.一般に船型デザイナー は、いわゆるV-型/U-型船尾フレームラインの最良 な妥協点を見出す作業を行う.V-型船尾はU-型船尾 に比べより弱い船尾縦渦を生成し,粘性抵抗もより 小さい.一方U-型船尾はより一様なプロペラ流入速 度分布を有し、プロペラ効率およびキャビテーション 特性について利点がある.船型デザイナーは、主機 配置などの幾何学的条件を含め、多くの設計要件を 同時に満足する形状を解とする形状最適化問題を解 くことになる.近年,タンカー船型設計支援を目的 とした計算流体力学(Computational Fluid Dynamics: CFD)手法が活発に開発され,特にレイノルズ平均ナ ビエ・ストークス方程式法(Reynolds-averaged Navier Stokes Equation Method: RaNS法)は,模型実験レベ ルレイノルズ数(Rn)の粘性流場予測に関し,ほぼ実用 段階に到達しつつある¹⁾⁻⁵⁾.

近年の計算機環境の急速な発達を背景として, RaNS 法の船体形状改良/最適化への応用も始まり, 実際 の設計現場における実用例も報告されている⁵⁾⁻⁷⁾. しかしながら, そのRaNS法の利用はいわゆる"Try and Error"的なものであり, 順問題解法に基づいた "Manual Optimization"の手段に止まっている. この 場合,設計パラメータの数が増加するほど最適解の決定

^{*1} 平成10年11月12日造船三学会秋季連合大会において 講演,原稿受付 平成10年12月1日

^{*2} 大阪府立大学工学部

^{*3} 川崎重工業(株)

が困難となり、また最適とされる解の精度にも限界があ ることを否めない. さらに、デザインサイクルの加速を 主目的として応用が進んだにも関わらず、デザインサイ クルが加速されるほど RaNS 法の Pre Processing(前処 理)および Post Processing(後処理)の頻度が増し,船 型デザイナーの付加的な労力が増加するに至っている. 一方, RaNS法を非線型計画法と結合し,設計パラ メータの感度解析に基づく船型最適化を行う手法は,上 述した"Manual Optimization"に対して"Automatic Optimization"とされるものであり、従来のRaNS法 の利用を今後大きく改善できると考えられる. CFDを 非線型計画法と結合する方法は,線形造波理論を用い た非粘性流法、もしくは境界層方程式法を用いた粘性 流法の応用において,これまでに比較的多くの研究が 行われてきた.しかし計算労力が極めて大きいRaNS 法を応用する方法は,最近の計算機能力の向上によっ てようやく実用可能段階になりつつある⁸⁾⁻¹⁶⁾.特に タンカー船型の船尾形状の最適化を試みる場合,境界 層が厚く発達する船尾近傍流場の解法としては RaNS 法が望ましい.しかしながら RaNS 法と非線型計画法 を応用した研究に関し、複雑な船尾形状を有するタン カー船型を主な対象とし, さらに得られた最適解を船 型デザイナーの従来の観点より詳細に評価した研究は まだ報告されていない.

本研究で開発した手法は、上述した RaNS 法と非線 形計画法を結合したものである.かつて著者らは同 様な手法の開発を試み,逐次線形計画法 (Successive Linear Programming: SLP)を適用して有望な結果を 得た11)-12).本研究ではその研究をさらに改良/拡張 し,特に船首バルブ・船尾バルブを有する近代的タン カー船型の船尾形状最適化を主目的とする. RaNS法 には近年著者らがタンカー船型を対象として開発し てきた有限解析法による解法を用い, また最適化ア ルゴリズムには逐次二次計画法 (Successive Quadratic Programming: SQP)を採用する.著者らは本研究に 関連する二次元翼最適化問題において、SQPはSLPに 比べより優れた収束性を有することを確認している ¹⁴⁾. 計算格子には, 従来多く用いられる H-O 型格子を 改良し、船首バルブへの適合性がより優れたC-O型格 子を採用する.また本研究では,比較的少ない設計パ ラメータ数でも妥当な規模で船型変更が可能となる形 状変更関数を提案する.本研究の第一報として,本論 文では特に粘性抵抗最小化を目指した船型改良の結果 について報告する.以下計算手法の概要を述べ,その 手法をSR221b類似船型に適用し、得られた粘性抵抗 最小船型の評価,また粘性抵抗低減の流体力学的要因, さらに従来の船型設計法の観点に基づく解の評価につ いて議論する.

2. 計算手法の概要

ここでは、本研究で用いた計算手法の概要を述べる. 流場の基礎方程式は、デカルト座標系(Fig.1)で記述さ れた、3次元非定常、非圧縮流体に関するRaNS方程式 である. 乱流モデルにはBaldwin-Lomax(BL)モデル ¹⁷⁾を用いる.BLモデルはプロペラ面内流場の予測精度 に関する問題点が指摘されているが、粘性抵抗の予測 に関する定性的信頼性があることは、これまでの研究 で確認されている^{4)-6),16),18),19)}.本研究では粘性抵抗 最小化のみを目的とするため、オリジナルのBLモデ ルを採用することとした.数値計算においては,基礎 方程式を数値的に生成した非直交曲線座標系に変換し, さらに輸送方程式を有限解析法により離散化する.速 度場/圧力場結合にはPISOタイプ/1ステップ法を採 用する.以上の方法により,(X,Y,Z)軸方向の平均流 速場(U,V,W),圧力場p,および渦動粘性係数vtが各計 算格子点で求まる.一方,計算格子には,本研究で対 象とした Blunt な船首を有するタンカー船型に対応し やすいC-O型トポロジーを採用する.まず初期船型の 計算格子を楕円型/代数型方程式法結合型の計算法で 生成し、最適化サイクルが進行し船体形状が変化する 度に代数型方程式法によって再生成する.数値計算法 の詳細、およびタンカー船型に適用した場合の精度評 価については、著者らの文献^{16),18),19)}を参照されたい.

最適化アルゴリズムには逐次二次計画法(SQP)を用 いる.著者らはSQPと逐次線形計画法(SLP)の特性評 価を行い,SQPはより高い収束性を有することを確認 している¹⁴⁾.なおSQPアルゴリズムには茨木ら²⁰⁾の 方法を用い,目的関数および制約条件の偏微分値は 差分法で求めた.最小化される目的関数は粘性抵抗 ($Cv \times WSA$)であり,これには形状変更による浸水表 面積(WSA)の変化も考慮されている.また粘性抵抗 の定義は,粘性抵抗($Cv \times WSA$)=粘性摩擦抵抗($Cvf \times WSA$)+粘性圧力抵抗($Cvp \times WSA$)である.計算



Fig. 1 Definition sketch of coordinate system

-258-

格子には108,000(=90×40×30,それぞれ長手,放射, ガース方向)格子および204,000(=170×40×30)格子 の2種を用いる.最適化計算は108,000格子で行い,解 として得られる設計パラメータを用いて変更船型を求 め,順問題的に108,000格子および204,000格子で検証 計算を行う.以下に示す抵抗値,粘性流場等は,全て 検証計算の結果である.文献^{6),16)}でに示したように, 108,000格子および204,000格子で予測された抵抗値順 位は一致しており,また実験値との比較で評価した定 量的予測精度は,204,000格子の方が高い.

つぎに船体形状の変更方法について述べる.非線型 計画法の計算労力は,形状変更関数に含まれる設計パ ラメータ数に大きく依存する.また物体を部分的に変 更する場合,変更対象領域の境界上における幾何学的 連続性が保証される形状変更関数が望ましい.本研究 では,比較的少ない設計パラメータ数で妥当な規模の 形状変更が可能な形状変更関数を検討した結果,以下 で述べる方法を採用することに決定した.まず船体形 状の変更は以下の式によって行う.

$$y(x,z) = y_0(x,z)B(x,z)$$
 (1)

ここで $y_0(x,z)$ は初期船型形状,B(x,z)は以下で定義 する形状変更関数である.

$$B(x,z) = 1 - B_1(z)sin(\pi a^3 b^3)$$
(2)

$$a = (x^{B_2(z)} - x_1^{B_2(z)}) / (x_2^{B_2(z)} - x_1^{B_2(z)})$$

$$b = (x^{B_2(z)} - x_2^{B_2(z)}) / (x_2^{B_2(z)} - x_1^{B_2(z)})$$
(3)

$$B_{1}(z) = f_{1}(z)\beta_{1} + \dots + f_{m}(z)\beta_{m}$$
$$B_{2}(z) = f_{m+1}(z)\beta_{m+1} + \dots + f_{n}(z)\beta_{n} \qquad (4)$$

ここで x_1 および x_2 は形状変更開始および終了境界のx座標, $B_1(z)$ および $B_2(z)$ はそれぞれ深さ方向および長 手方向制御関数, $\beta_1,\beta_2,...,\beta_m,\beta_{m+1},...,\beta_n$ は設計パラ メータ,さらに $f_1, f_2,..., f_m, f_{m+1},..., f_n$ はz方向に定義 した Cubic-Spline 補間関数である.また $B_1(z)$ および $B_2(z)$ は $z = z_{kcel} \sim 0$ で定義されており,つぎの端条 件を満たす.

$$\frac{\partial B_1}{\partial z} = \frac{\partial B_2}{\partial z} = 0 \qquad (z = z_{kcel}, z = 0) \qquad (5)$$

また関数B(x,z)は, x_1 および x_2 においてつぎの条件 を満たす.

$$B = 1$$
 , $\frac{\partial B}{\partial x} = \frac{\partial^2 B}{\partial x^2} = 0$ $(x = x_1, x = x_2)$ (6)

本形状変更関数は、形状変更領域の開始および終了境 界において、 $\partial y/\partial x$ および $\partial^2 y/\partial x^2$ が連続、すなわち 各 Waterline(WL)の勾配および曲率が自動的に連続す る性質を持つ.基本的には,船型デザイナーが形状を 部分的に変更しようとする際,変更開始および終了境 界におけるWLの連続性を考慮しつつ,各WLの一制 御点を摘み,幅方向に移動させる操作を想定したもの である.設計パラメータを適当に設定することで各 WLにおける制御点の位置,およびそこにおける移動 量を決定できる.本研究ではm = 3およびn = 6,す なわち設計パラメータ総数を6に設定した.

計算における設定レイノルズ数は $Rn=1.9 \times 10^6$,ま た設計パラメータの初期値はB(x,z) = 1,すなわち ($\beta_1,\beta_2,\beta_3,\beta_4,\beta_5,\beta_6$) =(0,0,0,1,1,1)とする.変更領域は 船体後半部(S.S.4~船体後端)に限定する.最適化の制 約条件は幾何形状に関するものとし,変更船型の排水 量(Disp.)は初期船型のもの以上,主機配置に関連し S.S.7/8で満たすべきフレームライン条件,最大喫水お よび最大半幅は初期船型のものと同一,さらにプロ ファイルは初期船型のものと同一とする.また各設計 パラメータの変動量が10⁻⁸以下に至った時点で収束と 見なし,最適化計算を終了する.

3. 計算結果

本研究では、SR229研究部会^{5),6)}で選択された基本 船型, SR221B類似船型の形状最適化を試みる.SR229 ではSR221B船型(*lcb=-2.6*)に加え,マニュアル的に形 状変更した3船型,すなわち*lcb=-3.1*,-3.6,-4.1船型 が準備された.粘性抵抗は*lcb*が小さくなるほど低下す る.本研究では*lcb=-3.6*船型を初期船型とし,前述し た計算手法と形状変更関数を用い,与えられた制約条 件のもとで粘性抵抗を最小化する船型を求める.Fig.2 に目的関数の収束状況を示す.



Fig. 2 Convergence history of objective function (Opt.SR221X-1: Cycle=3; Opt.SR221X-2: Cycle=7; and Opt.SR221X-3: Cycle=11(converged))

最適化サイクルの初期段階における減少が著し く、また最終段階における収束性も良い.以下の議

-259-

論において、lcb=-3.6船型をSR221X船型、また最適 化繰返し (Opt. Cycle)3回の解をOpt.SR221X-1船型、 7回の解をOpt.SR221X-2船型、さらに11回 (収束最 終回)の解をOpt.SR221X-3船型と定義し、最適化過 程の進行状況も検討する.Fig.3には各船型の浸水 表面積 (WSA)と排水量 (DISP.)、Fig.4には粘性抵抗 (= $Cv \times WSA = (Cvf + Cvp) \times WSA$)、またFig.5お よびFig.6にはそれぞれ粘性摩擦抵抗 ($Cvf \times WSA$)お よび粘性圧力抵抗 ($Cvp \times WSA$)を示し、以下の議論 で参照する.



Fig. 3 Comparison of computed wetted surface area (WSA) and displacement (DISP.)



Fig. 4 Comparison of computed viscous resistance $(=Cv \times WSA = (Cvf + Cvp) \times WSA)$



Fig. 5 Comparison of computed viscous-frictional resistance $(=Cvf \times WSA)$



Fig. 6 Comparison of computed viscous-pressure resistance $(=Cvp \times WSA)$

Opt.SR221X-1は最適化計算の収束途上の解であり, 最適化サイクル3回で得られたものである. Body Plan をFig.7に示す.

204,000 格子による粘性抵抗は,初期船型(SR221X) のものに対し約3.5%減,基本船型(SR221B)のものに 対し約5%減を示す(Fig.4). この段階では船型にナッ クル的なビルジ形状が見られる.限界流線(Frictional streamlines)を調べた結果、ナックル部が限界流線に ほぼ沿った形で形成されており、抵抗増加をもたらす 3次元剥離の増加などがないことを確認した.最適化 途上の解であるが、ナックル的ビルジを有しつつも抵 抗減となった船型として興味深い. 最適化サイクルの 初期段階においてこの形状が得られた理由は、粘性抵 抗減少の観点から設計パラメータの感度解析を行った 結果、まず船尾ビルジ近傍の体積分布を静止水面船尾 近傍に比較的大規模に移動させる操作, すなわち初期 船型の船尾ビルジ近傍に見られる低圧力領域(Pressure pocket)の平坦化が最も有効と判断されたためである. この操作は船尾ビルジ近傍の三次元剥離を減少させ, 船尾縦渦の強度を弱める効果があり、結果として主に



Fig. 7 Comparison of body plan between the original (SR221X) and modified (Opt.SR221X-1) hull forms

粘性圧力抵抗の減少が達成できている(Fig.6).

Opt.SR221X-2は最適化サイクル7回で得られたも のであり、204,000格子による粘性抵抗は、初期船型 (SR221X)のものに対し約4.5%減,基本船型(SR221B) のものに対し約6%減を示す(Fig.4). Fig.8に制約条件 を課したS.S.7/8におけるフレームラインの比較、さ らにFig.9およびFig.10にこの船型と初期船型の断面 積分布の差,および形状の比較を示す.



Fig. 8 Comparison of frame lines at S.S.7/8 for Opt.SR221X-2



Fig. 9 Differences in cross-sectional area between the original (SR221X) and modified (Opt.SR221X-2) hull forms

先に述べたS.S.7/8のフレームライン制約条件は満 足されており、またOpt.SR221X-1で現われたナック ル的なビルジ形状が滑らかに変更され、より実用的な 船型になっている.S.S.7/8における幅方向の変更傾 向はビルジ近傍の減少、および静止水面近傍の増加で あり、ビルジ近傍の変化がまず制約条件によって拘束 されたことがわかる.体積分布はS.S.11/2近傍前方で 減少し、その後方で増加している.すなわち初期船型 の*lcb*がより後方に移動したことになる.

Opt.SR221X-3は最適化サイクル11回で最適化計算



Fig. 10 Comparison of buttock and water lines, and body plan between the original (SR221X) and modified (Opt.SR221X-2) hull forms

の収束解として得られた.粘性抵抗はOpt.SR221X-2の ものよりやや低く,形状もOpt.SR221X-2とほとんど差 がない(Fig.4).しかしながら,排水量がOpt.SR221X-2 のものよりやや減少し,結果的に初期船型(SR221X) のものより0.1%程度小さくなった(Fig.3).これは数値 誤差の範囲で排水量制約条件が満足されたためである. 本最適化手法がOpt.SR221X-2よりさらに低粘性抵抗 を目指した際,排水量をやや減少させた分だけ抵抗減 になったものと考えられる.

Opt.SR221X-1~Opt.SR221X-3全ての変更船型に関 し,S.S.7/8におけるフレームライン条件および他の幾 何学的制約条件は,数値誤差の範囲で全て満足され ている.また改良船型の表面近傍流場を検討した結 果,初期船型(SR221X)の船尾近傍ビルジ部で見られ る Pressure pocketが平坦化され,それにともなって近 傍流場の三次元剥離が明らかに減少していることがわ かった (Fig.11).



Fig. 11 Comparison of surface pressure contours and frictional streamlines between the original (SR221X) and modified (Opt.SR221X-2) hull forms



Fig. 12 Iso-axial vorticity (ω_1) surfaces near the stern for the original (SR221X) and modified (Opt.SR221X-2) hull forms

さらに粘性抵抗の減少は,主に圧力抵抗の減少に よって達成されており,これは上記三次元剥離の減少 に関連する渦抵抗減少も含まれる(Fig.4-6). Fig.12に は船尾縦渦強度の等値面の比較を示す.

初期船型にくらべ,改良船型の船尾縦渦は大幅に減 少していることがわかる.一方,全ての変更船型の体 積分布は,初期船型(SR221X)のものより船尾方向に 移動した形になっている.フレームライン形状は概ね V型に移行しているが,上記体積分布の変更と相まっ て,単純なV型変形ではないことがわかる.船体後半 船底部の体積がより圧力の高い船尾水線面近傍に移動 し,これによって圧力抵抗減少が達成できた事実にも 着目すべきであろう.

4. 従来の船型設計法の観点に基づく解の評価

従来の船型設計のプロセスは,船型計画ステージ と性能設計ステージに大別できる.前者で船体主要 目(L,B,D,d,Cb,Cm,lcb)を設計変数とし,DWTと サービス船速要求を制約条件として目的関数であるコ スト(本来はライフサイクルコストであるべきである が,通常は建造コストが対象となる)を最小化する. その一般的手順は所謂Type Ship方式であり,経験的 にType Ship 候補選定を行ない,その類似船型につい ての性能データベースから回帰分析的に設計変数に対 する性能評価を算定し,直接探索的に最適解を求めて いる.一方,後者の性能設計ステージでは,前者で決 定された要目を基準とし,改良性能を有する船体形状 (線図)を求めることになる.

本報は後者のステージに関するものであり,ここに おける基本的な目標は,本来はコストの最小化である. 機能,工作,性能の多面的制約条件を考慮した上で, 更なるコスト削減が図られ,その内の性能評価過程に おいて本報の粘性抵抗最小化ステップが位置づけられ る.このステップでも,従来法では一般にType Ship 性能データベースを基に,順問題的に直接探索法で 最適化を図るが,その際*Cp*カーブ(Run Length,肩 部曲率,後短部*Cp*値),WLカーブ(*Cp*カーブと同 形式),船尾部代表フレームライン形状(今の場合, S.S.7/8に相当)が主たる設計変数となる.

順問題解法を基本とする従来法では、選定 Type Ship の線図から通常 Cp カーブ,WL カーブの変形評価から 設計に入り、回帰分析からは Run Length をより長く、 肩落ち型が有利となるのが一般的である.また、船尾 端、すなわちトランサム深度は主船体部粘性抵抗と トランサム部剥離抵抗の和を最小にするべく決めら れ、現在のところ実験により設定された設計基準に より決定されているようである.つぎにフレームラ インの設計に入るが、従来法である Type Ship 法では Type Shipの設計思想を踏襲する関係上、Type Ship のフレームラインの特徴を維持した範囲での変形評価 となる.その際、粘性圧力抵抗を2次元的な境界層発 達による運動量欠損と、ビルジ渦(3次元剥離)による 誘導抵抗の和として捕らえ、その最小化を図ることを 目標として、フレームライン変形を行なっている.

本報の計算で得られた改良船型は,初期船型(SR221X) と比べて V 型となり、渦抵抗を減ずる方向に変更され、 また船底側体積が船尾水線面近傍へ移動しているが、 これは従来法の考え方と一致すると言える. 初期船型 はビルジ曲率の大きい部分が船尾全域に分布し、吃水 中間レベルから上は2次元的船型の傾向がある.この 様な船型では粘性圧力抵抗が大きくなる可能性があり、 改善の余地がある. 改善方向としては本報の計算結果 に見られるように船底側体積を船尾水線面近傍へ移動 し、それによってWL方向に2次元的な流れを3次元的 あるいはバトックフロー的にし、圧力勾配を緩和させ るべきである.また同時に後端部に見られる球状船尾 的形状を付加することで適度なビルジ渦を生成させ, 外部流を船体側に誘導する作用で,スクリューアパー チャー部によく見られる境界層の肥大化を抑制させる 措置がとられるべきであろう.本報の計算では、この

一連の改善を自動的に実行したものであると言える.

5. 結言

本研究は,RaNS方程式法に基づくCFD手法と非線 形計画法の応用による,タンカー船型の船尾形状最適 化に関するものである.本第一報では,特に粘性抵抗 最小化を目指した船型改良の結果について報告し,粘 性抵抗低減の流体力学的要因について,さらに従来の 船型設計法の観点に基づいた解の評価について議論し た.その結果,本研究で開発した手法は,今回考慮し た形状変更関数および制約条件のもとで,最小粘性抵 抗を有するタンカー船型船尾形状を自動的に得ること ができ,さらにその形状変更傾向は従来の設計手法で 採用される方法と合致したものであることが明らかと なった.今後より実際的な目的関数の考慮,例えば粘 性抵抗/伴流情報も包括した目的関数を最小化する手 法へ拡張する必要がある.

なお本研究の一部は(社)日本造船研究協会(SR229研 究部会)との共同研究として,さらに日本学術振興会補 助・日米科学協力事業および米国National Foundation of Science (Contract INT-9513138)補助のもとに行わ れました.委員各位ならびに関係各位の御助言・御協 力に謝意を表します.

参考文献

- 日夏宗彦,日野孝則:船体周りの流れの計算法, 船体周りの流れと船型開発に関するシンポジウム, April 1993, pp.225-262
- Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994
- 3) Stern, F., Paterson, E.G. and Tahara, Y.: CFDSHO-IOWA: Computational Fluid Dynamics Method for Surface-Ship Boundary Layers, Wakes, and Wave Fields, IIHR Report, No. 381, Iowa Institute of Hydraulic Research, Iowa City, IA 52242, USA, 1995
- 日本造船研究協会:第222研究部会「大型肥大船船 尾流場推定法の高度化」報告書(第3年度),1996
- 5) 日本造船研究協会: 第229研究部会「数値流体力 学による最適船型設計法の研究大」平成8年度報 告書, 1997
- 6) 日本造船研究協会:第229研究部会「数値流体力 学による最適船型設計法の研究大」平成9年度報 告書,1998
- 7) Larsson, L., Kim, K.J., Esping, B. and Holm, D.: Hydrodynamic Optimization Using SHIPFLOW, Proc. The 5th International Symposium on Practical Design in Shipbuilding (PRADS'92), Newcastle, 1992

- Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998
- 9) 鈴木和夫,小柴幸雄:船型計画法と線図生成,船 体周りの流れと船型開発に関するシンポジウム, April 1993, pp.163-201
- 10) JANSON, C.E., Kim, K.J., and Larson, L. : Optimization of the Series 60 Hull a Resistance Point of View, Proc. CFD Workshop Tokyo 1994, Tokyo, March 1994
- 11) 浜崎準一,姫野洋司,田原裕介:非線形計画法による 船型最適化の試み(第3報)-粘性抵抗最小化を 目指した船体後半部形状の改良-,関西造船協会 誌,第225号,1996, pp.1-6
- 12) 浜崎準一,姫野洋司,田原裕介:非線形計画法による船型最適化の試み(第4報)-伴流と粘性抵抗 を考慮した船体後半部形状の改良-,関西造船協会 誌,第226号,1996,pp.15-21
- 日野孝則:Fluid-dynamic Shape Optimization Using Sensitivity Analysis of Navier-Stokes Solutions, 関西造船協会誌, 第226号,1996,pp.49-54
- 14)田原裕介,姫野洋司:CFDによる二次元翼形状 改良問題に関する研究,関西造船協会誌,第229 号,1998,pp.27-35
- 15) Hino, T., Kodama, Y., and Hirata, N.: Hydrodynamic Shape Optimization of Ship Hull Forms Using CFD, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 533-541
- 16) Tahara, Y., Himeno, Y., and Tsukahara, T.: An Application of Computational Fluid Dynamics to Tanker Hull Form Optimization Problem, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 515-531
- 17) Baldwin, B.S. and Lomax, H.: Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, 1978, pp. 1-8
- 18) Tahara, Y. and Himeno Y.: Application of Isotropic and Anisotropic Turbulence Models to Ship Flow Computation, J. Kansai Society of Naval Architects, Japan, No. 225, 1996, pp. 75-91
- 19) Tahara, Y. and Himeno, Y.: Computation of Isotropic and Isotropic Turbulent Flows around Tanker Forms, Proc. 3rd Korea-Japan Joint

Workshop on Ship and Marine Hydrodynamics, Daejon, Korea, 1996, pp.191-198

 20) 茨木俊秀,福島雅夫:FORTRAN77 最適化プログ ラミング,岩波書店,1991,pp.167-207

CFDによるタンカー船型の船尾形状最適化 (第2報)-伝達馬力最小化-*¹

正会員田原裕介*2,正会員齋藤泰夫*3 正会員松山博志*4,正会員姫野洋司*2

CFD-Aided Optimization of Tanker Stern Form (2nd Report) - Minimization of Delivered Horse Power -

By Yusuke TAHARA (Member), Yasuo SAITOH (Member) Hiroshi MATSUYAMA (Member) and Yoji HIMENO (Member)

This paper presents optimization of tanker stern form based on viscous flow information obtained from Computational Fluid Dynamics. In this 2nd report, main focus is placed on minimization of delivered horse power at constant ship speed originally designed, and it is shown that a single objective function proposed in the present study is capable for it. The numerical method is based on extension and modification of the method presented in the 1st report on minimization of viscous resistance, i.e., coupling of Reynolds-averaged Navier-Stokes (RaNS) equation solver and successive quadratic programming. Accuracy of RaNS solver in prediction of near wake field is improved by modification of turbulence model. Constraints considered in the present study are basically same as those used in the 1st report, i.e., geometrical requirements of optimized hull for displacement, profile, maximum beam and depth, and stern frame line to secure enough space for engine room. In conclusion, the present method appeared to successfully optimize the given tanker stern form, and the modification trends automatically demonstrated in the present work agree well with those commonly in use in traditional tanker hull form design.

Keywords : Tanker Hull Form Optimization, Reynolds-averaged Navier-Stokes Equations, Successive Quadratic Programming, Minimization of Delivered Horse Power

1. 緒言

近年急速に発達した計算流体力学(Computational Fluid Dynamics: CFD)技術を背景として,レイノルズ 平均ナビエ・ストークス方程式法(Reynolds-averaged Navier Stokes Equation Method: RaNS法)のタンカー 船型船尾形状改良/最適化への応用も始まり,実際の 設計現場における実用例も報告されている¹⁾⁻⁴⁾.順問 題解法を拡張した"Manual Optimization"だけでな く,設計パラメータの感度解析に基づく"Automatic Optimization(自動最適化)"の研究も始まった.特に後

*4 ヤンマーディーゼル(株),研究当時大阪府立大学大学院

者は従来のRaNS法の利用を今後大きく改善できると 考えられ、その現状は、粘性抵抗を目的関数とした場 合において、船型デザイナーが従来用いてきた船型改 良法と合致した傾向を自動的に再現できる段階にある.

一方,推進効率向上を目指した船尾形状改良/最適 化の際,粘性抵抗以外の情報,すなわちプロペラ面内 流場情報やプロペラ効率などを考慮した上で最適解を 求める必要がある.この場合,対象とするタンカー 船型が一定速度で航行すると仮定すれば,推進効率 向上に直接関連する最小化目的関数としては伝達馬 力(Delivered Horse Power: DHP)が最も有効である. DHP最小化を目指した船型最適化手法に関する研究 はこれまでに報告されていない.かつて著者らは粘性 抵抗およびプロペラ面内の一指定点流速を同時に考慮 した船型最適化を試み,開発した計算手法が有効に機

^{*1} 平成11年5月21日関西造船協会春季講演会において講 演,原稿受付 平成11年6月10日

^{*2} 大阪府立大学工学部

^{*3} 川崎重工業(株)

能することを示した⁵⁾.しかしその研究は*DHP*最小 化を目指したものではなく,また使用された乱流モデ ルの限界に起因する伴流計算精度の問題点などを解決 するには至らなかった.

本研究では、RaNS法と逐次二次計画法(Successive Quadratic Programming: SQP)を結合し、特に近代タ ンカー船型の船尾形状最適化を行う計算手法の開発を 行っている.著者らは本研究に関連する前報⁴⁾におい て粘性抵抗最小化に関する研究結果を報告し、計算手 法のカーネル部が有効に機能することを示した.本報 の主眼は、一定計画速度における DHP 最小化を目指 した船尾形状最適化である.計算手法は,基本的に前 報の方法を拡張/改良したものである.RaNS法には近 年著者らがタンカー船型を対象として開発してきた方 法を用い、今回乱流モデルを改良することによってプ ロペラ面内の平均流速場計算精度を向上させた.本論 文においては、まず一定計画速度 DHP 最小化が抵抗 と伴流値を変数とした単一目的関数の問題に帰着する ことを示し、粘性抵抗最小化および DHP 最小化に関 する数値計算結果を検討して手法の妥当性を評価する. さらに従来の船型設計法の観点に基づいた最適解の評 価を行い,本研究で提案する手法は従来型設計法と合 致した傾向を自動的に再現できることを示す.加えて, 一定計画速度 DHP 最小化による船型は,粘性抵抗最 小化では得られない特徴を有することが確認された.

2. 計算手法の概要

2.1 最小化目的関数の設定

ここでは最小化される目的関数について述べる.ま ず伝達馬力の定義に従い,DHPを以下のように表現 する.

$$DHP \propto \frac{RV_s}{\eta_p} \tag{1}$$

ここで $R = Cv \times WSA = (Cvp + Cvf) \times WSA$ は粘 性抵抗, V_s は航行速度, η_p はプロペラ効率である. 形 状変更の際, V_s , 推力減少率1-t, プロペラ効率比 η_R を一定とし, $1-w_e$ (有効伴流係数) $\propto 1-w_n$ (= W_n :公 称伴流係数)および前進率 $J \propto W_n/nD(n:$ プロペラ回 転数,D:プロペラ直径)と仮定, さらにエンジンはトル クー定制御とする. さらに諸量の局所線形性を仮定し, 変数の増分に Δ を付けると, (1)式は次式のように表 わされる.

$$\frac{\Delta DHP}{DHP} = \frac{\Delta R}{R} - \frac{\Delta \eta_p}{\eta_p} \tag{2}$$

一方, np および推力Tは以下のように定義されている.

$$\eta_p = \frac{K_t}{K_q} \frac{J}{2\pi} \frac{1-t}{1-w_e} \eta_R \tag{3}$$

$$T = \rho n^2 D^4 K_t = \frac{R}{1-t} \tag{4}$$

ここで K_t はスラスト係数, K_q はトルク係数,そして ρ は流体密度である.(2)式の導出と同様, η_p およびRについて以下の式を得る.

$$\frac{\Delta \eta_p}{\eta_p} = \frac{\Delta K_t}{K_t} + \frac{\Delta J}{J} - \frac{\Delta K_q}{K_q} - \frac{\Delta W_n}{W_n} \tag{5}$$

$$\frac{\Delta R}{R} = \frac{\Delta K_t}{K_t} + \frac{2\Delta n}{n} + \frac{4\Delta D}{D} \tag{6}$$

つづいてJについて(6)式を考慮することにより

$$\frac{\Delta J}{J} = \frac{\Delta W_n}{W_n} - \frac{\Delta n}{n} - \frac{\Delta D}{D}$$
$$= \frac{\Delta W_n}{W_n} - \frac{1}{4} \left(\frac{\Delta R}{R} - \frac{\Delta K_t}{K_t} + \frac{2\Delta n}{n}\right)$$
$$= \frac{1}{1 - \alpha_t/4} \left(\frac{\Delta W_n}{W_n} - \frac{\Delta R}{4R} - \frac{\Delta n}{2n}\right)$$
(7)

このとき、 α_t は設計経験値に基づく定数 α_t とし、次のように定義されている.

$$\alpha_t = \frac{\Delta K_t}{\Delta J} \frac{J}{K_t} \tag{8}$$

次に設計経験定数α_gを以下のように定義し

$$\alpha_q = \frac{\Delta K_q}{\Delta J} \frac{J}{K_q} \tag{9}$$

さらに(2)(5)(7)式を纏めることにより, DHPに関する以下の関係式が導かれる.

$$\frac{\Delta DHP}{DHP} = \frac{\Delta R}{R} - \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4} \left(\frac{\Delta W_n}{W_n} - \frac{\Delta R}{4R} - \frac{\Delta n}{2n}\right) + \frac{\Delta W_n}{W_n}$$
(10)

ここでトルクー定制御の仮定に基づき,回転数に関す る項をDHPで置き換えると

$$\frac{\Delta DHP}{DHP} = \left(1 + \frac{1}{4} \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right) \frac{\Delta R}{R} + \left(1 - \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right) \frac{\Delta W_n}{W_n} + \frac{1}{2} \left(\frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right) \frac{\Delta DHP}{DHP}$$
(11)

さらに係数を整理することによって

$$\frac{\Delta DHP}{DHP} = \frac{\alpha_1}{\alpha_0} \frac{\Delta R}{R} + \frac{\alpha_2}{\alpha_0} \frac{\Delta W_n}{W_n}$$
(12)

$$\alpha_0 = 1 - \frac{1}{2} \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}$$
(13)

$$\alpha_1 = 1 + \frac{1}{4} \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4} \tag{14}$$

$$\alpha_2 = 1 - \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4} \tag{15}$$

よって(12)式より得られる関係式は

$$DHP \propto R^{\alpha 1/\alpha 0} W_n^{\alpha 2/\alpha 0} \tag{16}$$

もしくは

$$DHP^{\alpha_0/\alpha_1} \propto RW_n^{\alpha_2/\alpha_1} = RW_n^{\alpha} \tag{17}$$

となる.上述した仮定に基づき一定計画速度にお けるDHP最小化を考える場合,上式中の定数が 設計経験値で与えられれば,DHP最小化問題が 抵抗と伴流値を変数とした単一目的関数を用いた 最適化問題に帰着することになる.実際に肥大船 型の設計経験値では,例えば $(\alpha_t, \alpha_q, \alpha_0, \alpha_1, \alpha_2) = (-0.5148, -0.2686, 0.6661, 1.1670, 0.3322)$ と設定でき,従っ て $\alpha = \alpha_2/\alpha_1 = 0.285$ となる.そこで本研究では,最適 化目的関数Fを以下のように設定する.

$$F = RW_n^{\alpha} \tag{18}$$

一方, DHPの推定値を初期船型のものと比較する際, その減少した割合に着目するとすれば, (16)式が使用 できる.

2.2 修正BLモデルの採用

ここでは、本研究で用いた修正 Baldwin-Lomax(BL) 乱流モデルについて述べる。BLモデルの原型⁶⁾では、 壁面境界層を内層と外層にわけ、各層での渦動粘性係 数 ν_t を以下のようにして求める。

Inner layer: $\nu_t = l^2 |\omega|$ (19)

 $Outer \quad layer: \nu_t = kC_{CP}F_{Wake}F_{Kleb} \tag{20}$

ここで、*l*は混合長 (= $\kappa y [1 - exp(-y^+/A^+)]$), $\kappa = 0.4$ はカルマン定数、*y* は壁面からの垂直距離、 $y^+ = y R_n \sqrt{\tau_w}$ は 無 次元 距離、 R_n は レ イ ノ ル ズ 数、 τ_w は 壁 面 摩 擦 応力、 $|\omega| = \sqrt{\omega_x^2 + \omega_y^2 + \omega_z^2}$)は渦度ベクトルの絶対 値、k = 0.0168は Clauser 定数、 F_{Wake} は関数 $F_{Wake} = min(y_{Max}F_{Max}, C_{WK}y_{Max}U_{Dif}^2/F_{Max})$ で与えられる 値、 F_{Max} は関数 $F(y) = y |\omega| [1 - exp(-y^+/A^+)]$ の最大 値、 y_{Max} は関数F(y)が最大値をとるときの y、 $F_{Kleb} = [1 + 5.5(C_{kleb}y/y_{Max})^6]^{-1}$)は Klebanoff の 間 欠 係 数、 U_{Dif} は表面法線方向速度プロファイルより与えられる次式の値 $U_{Dif} = (\sqrt{u^2 + v^2 + w^2})_{Max} - (\sqrt{u^2 + v^2 + w^2})_{Min}$, そして $A^+ = 26$, $C_{CP} = 1.6$, $C_{WK} = 1.0$, $C_{Kleb} = 0.3$ はモデル定数である.モデル定数は ν_t が内層から外層 にかけてスムーズに連続し,かつ平板境界層の速度分 布が再現できるように設定されている.

修正BLモデルは、外層における圧力勾配影響および縦渦の存在による渦動粘性係数ν_tの減衰を考慮する ものであり、その基本型はSR222研究部会⁷⁾において 詳細に検討された.このモデルは、特に平均速度場の 予測に関し、BLモデルの精度向上を目指して開発され たもので、工学的利用においては有効であることが確 認されている^{7),8)}.また修正BLモデルは、他の高次乱 流モデルに比べてより簡便で、さらに高い計算安定性 を有する利点がある.これらの理由により、本研究で は修正BLモデルを採用した.

修正BLモデルでは外層における ν_t を以下のように与える.

$$\nu_t = k C_{CP}^* F_{Wake} F_{Kleb} C_{DM} \tag{21}$$

$$C_{cp}^* = C_{CP} \left\{ 1 - \tanh[\beta_{PG} \frac{y_{Max}}{\rho F_{Max}} \frac{\nabla P \cdot \vec{u}}{|\vec{u}|}] \right\}$$
(22)

$$C_{DM} = 1 - \frac{|\vec{u} \cdot \vec{w}|}{|\vec{u}| |\vec{w}|} = 1 - |\cos(\theta)|$$
(23)

ここで 祉は流速ベクトル, ω は渦度ベクトル, θ は iと ω の偏角, ∇P は圧力勾配ベクトル, $\beta_{PG} = 50$ はモデ ル定数である.上述したSR222研究部会で検討された 修正モデルでは, 内層の $|\omega| \ge |\omega_t| = |(\vec{n} \times \vec{\tau_w}) \cdot \omega| / |\vec{\tau_w}|$ $|(ここで \vec{n}$ は船体表面単位法線ベクトル, $\vec{\tau_w}$ は摩擦応 力ベクトル)で置き換え,内層においても ν_t を強制減 衰させている.しかしながら,タンカー船型のモデル スケール計算においては,特に船尾端近傍においてバ ブル型剥離が存在することが多く,その剥離領域で $|\omega_t|$ | を用いることは原理的に適切ではない.従って本研究では $|\omega|$ の与え方はBLモデル原型のままとした.さら にSR196船型やSR221船型に関する計算値と実験値を 比較してモデル定数の再評価を行った結果, $\beta_{PG}=50$ が最も有効であるという結論を得た.

2.3 数値計算法の概要

ここでは、本研究で用いた数値計算法の概要を述べ る.本研究で用いた計算手法は、乱流モデルの改良や プロペラDISC面における流場補間および積分機能の 追加を除き、基本的に前報⁴⁾のものと同一である.流 場の基礎方程式は、デカルト座標系(Fig.1)で記述され た、3次元非定常、非圧縮流体に関するRaNS方程式で ある.数値計算においては、基礎方程式を数値的に生 成した非直交曲線座標系に変換し、さらに輸送方程式

-267-

を有限解析法により離散化する.速度場/圧力場結合に はPISOタイプ/1ステップ法を採用する.以上の方法 により,(X,Y,Z)軸方向の平均流速場(U,V,W),圧力 場p,および渦動粘性係数 ν_t が各計算格子点で求まる. 一方,計算格子には、本研究で対象としたBluntな船 首を有するタンカー船型に対応しやすいC-O型トポロ ジーを採用する.まず初期船型の計算格子を楕円型/ 代数型方程式法結合型の計算法で生成し、最適化サイ クルが進行し船体形状が変化する度に代数型方程式法 によって再生成する.数値計算法の詳細、およびタン カー船型に適用した場合の精度評価については、著者 らの文献^{1),3),4)}を参照されたい.乱流モデルには上述 した修正BLモデルを β_{PG} =50で用いる.

また本研究で行う DHP 最小化では、上述した目的 関数を評価するために公称伴流係数 ($W_n = 1 - w_n$)を 計算する必要がある.本研究では、Fig.2に示すよう なプロペラ DISC計算面 (格子分布:周方向50×放射 方向50)を設定し、粘性流場計算格子上で得られる流 場情報を 2次元補間で転送することによって W_n の計 算を行った.プロペラ DISC計算面の格子に関しては、 上述した格子数を倍にしても積分値への影響が 0.001 %程度であることを確認したため、今回設定した格子 数で十分な精度があると判断した.今回の最適化初期 船型は、前報でも採用した SR221X 船型 (SR229 研究部 会³⁾においては SR221B-L2 船型と定義)であり、今回 使用したプロペラ位置や直径などの幾何情報について は、SR229 研究部会で標準的とされたものを使用した.

本研究で採用したC-O型計算格子, 6パラメータ形 状変更関数, Initial unity 初期形状変更関数值, SQP型 最適化アルゴリズムおよび収束判定条件(各設計パラ メータの変動量が10-8以下に至れば計算を終了),プロ ファイルー定条件, 排水量(Disp.)制約条件(排水量は 初期船型のもの以上), 主機配置要件に基づくS.S.7/8 フレームライン制約条件,最大喫水および最大半幅 一定制約条件,形状変更領域(S.S.4~船体後端),さら に設定レイノルズ数($Rn=U_{\infty}L/\nu=1.9 \times 10^{6}, U_{\infty}, L, \nu$) はそれぞれ船速,船長,流体の動粘性係数)は,前報 4)で使用したものと同一である.加えて前報と同様, 最適化計算は粗計算格子(90×40×30=108,000,それ ぞれ長手,放射,ガース方向)で行い,得られた船型を 用いて密計算格子 (170×40×30=204,000)を生成し, 後者の結果を用いて流場情報の解析を行う. 今回乱流 モデルを改良した際、粗および密計算格子間の抵抗値 順位,伴流係数順位,(16)式による推定DHP順位は 同一であることを確認した. さらにこれまでに行った 研究において, 密格子による計算結果の定量的信頼性 は、モデルスケール Rnに関する限り実用的レベルで あることを確認している1),3),4).



Fig. 1 Definition sketch of coordinate system



grid at propeller section

以下の議論において、計算結果は設定一様流速 U_{∞} , 船長Lで無次元化して示す.なおX=0,1はそれぞれ FP,APに対応し、またDHPの推定には(16)式を(α_0 , α_1 , α_2)=(0.6661,1.1670,0.3322)で用いていることに注 意されたい.

3. 計算結果

3.1 経験定数αの設定

本研究では、目的関数に含まれる経験定数 α の影響 を調査するために、 $\alpha=0$, 0.2, 0.3の3ケースの船型 最適化を検討した.前述したとおり、肥大船型の設計 経験値の α は約0.3である.一方、 $\alpha=0$ の場合は粘性 抵抗最小化と同義になる.ここで $\alpha=0$, 0.2, 0.3の最 適解をそれぞれ Optimized Case W00, W02, W03と 定義し、以下の議論で参照する.収束までに要した最 適化サイクルはそれぞれ 30,9,16回であった.また全て の最適解に関し、S.S.7/8におけるフレームライン条件



Fig. 3 Comparison of computed nominal wake coefficient $(W_n=1-w_n)$ and estimated delivered horse power (DHP: evaluated by eq. (6) with assumed $(\alpha_0, \alpha_1, \alpha_2)=(0.6661, 1.1670, 0.3322))$



およびその他の幾何学的制約条件は,数値誤差の範囲 で全て満足されていることを確認した.

3.2 最適解の積分値の傾向

ここでは、 α の変化が最適解の伴流係数や抵抗値 などの積分値へ与える影響に関し、計算結果を検討 する.まず解の傾向を述べ、つづいて抵抗と伴流情 報の関連、そして今回設定した目的関数の妥当性に ついて議論する.Fig.3は伴流係数(W_n)および伝達 馬力(DHP)の比較を示す.またFig.4には粘性抵抗 ($=Cv \times WSA = (Cvf + Cvp) \times WSA$)の比較を示し、 その内訳である粘性摩擦抵抗($Cvf \times WSA$)および粘性 圧力抵抗($Cvp \times WSA$)の比較をFig.5に示す.さらに 排水量DISP.および浸水表面積WSAの比較をFig.6 に示し、それらを以下の議論で参照する.

前述したように、Case W00(α =0)は粘性抵抗最小 化の解であり、最適化の設定条件は前報⁴⁾と全て同じ である.今回は前報と異なる乱流モデルを使用して いるが、Case W00の粘性抵抗減少および船尾形状は 前報で示した最終収束解のものとほぼ同じであった.



これは、前報で使用した Original BLモデルは、抵抗 値順位の推定に関し、今回使用した改良 BLモデルと 同等の精度を有するためである. Case W00の段階で、 初期船型のものと比較した粘性抵抗は約6.4%減少、 そして推定 DHP は約17%減少している.

一方, α ≠ 0とした場合, すなわち最小化目的関数 に伴流情報を考慮した場合は, さらに推定 DHP が減 少する傾向がある. Case W02 ($\alpha = 0.2$)のDHPは初 期船型値比較でW00のものより約3.4%減少し, Case W03 ($\alpha = 0.3$)においてはさらに約1.2%減少してい る. 最適解の W_n はDHPと同傾向を示し, Case W00, $W02, W03の順に低い値となっている. これは<math>\alpha$ が大 きいほど,目的関数減少の際にW_nを減少させる比重 が増加するためである.また最適解のWnの減少が DHP減少に至るのは、(5)式の関係によりプロペラ効 率が増加するためである.他方,最適解の粘性抵抗は W_n やDHPと逆傾向を示している. Fig.5に示される ように粘性抵抗の増減に関しては粘性圧力抵抗の影響 が支配的である、これは Case W00, W02, W03の順に 船尾縦渦強度が増加することと関係があり、その傾向 は後述する伴流分布において明示されている.

最適解の排水量に関しては, Case W00が初期船型



rig. 8 Distribution of cross-sectional area (CP curve) for the original and optimized hull forms

と同等であるのに対し, $\alpha \neq 0$ の場合はむしろ増加させる傾向がある.なおCase W00の排水量が初期船型のものより約0.1%減少しているのは,数値誤差の範囲で排水量制約条件が満たされたためである.Case W02においては初期船型値比較で約0.6%増加し,またCase W03においてはさらに約0.4%増加した.一般に粘性抵抗最小化を目指した最適化計算では,排水量は制約条件で制限されるまで低下する傾向がある.今回の $\alpha \neq 0$ では粘性抵抗最小化とは異なる傾向が見られるが,その原因は W_n 減少を促進するために船尾端近傍で体積分布増加が起こったためであり,その詳細は後述する.

つぎに今回使用した目的関数の妥当性を検討する. 粘性抵抗低減化が図られた船型においては, W_nの減少



and body plan between the original (S-R221X) and optimized (Case W00) hull forms

は通常粘性抵抗増加をもたらすことになる。肥大船型 に関し、 W_n 減少と抵抗増加の度合いをどの程度の比率 でDHPに関連づけるかに関する研究は、これまでに 報告されていない、本研究で用いた目的関数によれば、

$$\frac{\Delta F}{F} = \frac{\Delta R}{R} + \alpha \frac{\Delta W_n}{W_n} \tag{24}$$

となり、これは目的関数の増減率と粘性抵抗および伴 流係数の増減率の関係を示している. Case W03のよ うに α =0.3とし、仮に目的関数の増減が無かった場合、 W_n の減少が 10^{-1} のオーダーであったとすると、それ にバランスする抵抗増加は 10^{-2} のオーダーである. こ れは形状影響係数1+kが渦抵抗の増減によって変化す るオーダーであり、オーダー的観点でも本目的関数は 妥当な相関関係を与えている. 今回の α =0.3の計算結 果では、 α =0の結果より粘性抵抗は増加したが W_n は 減少し、結果的には α =0の結果よりも推定DHPが減 少していることから、DHP最小化という観点で船型 最適化を行う場合は、今回行ったように粘性抵抗およ び伴流情報を包括的に考慮した目的関数を用いる必要 があると考えられる.



Fig. 10 Comparison of buttock and water lines, and body plan between the original (S-R221X) and optimized (Case W03) hull forms

3.3 最適解の形状および伴流分布の傾向

ここでは最適解の船型形状を検討する.まずFig.7に 長手方向断面積分布の最適化船型と初期船型との差を 示す. Case W00ではSS.2 1/2(X=0.75)近傍で体積分 布が減少し,より後方で増加することにより,全体とし て体積分布が後方に移動している.また全体の排水量 を初期船型のものとほぼ同値に抑えるため,船尾端近 傍の体積増加もそれ以前の場所で起こった体積減少と 釣合う形になっている.一方,Case W02ではSS.2 1/2 近傍の体積減少が抑えられ,さらに船尾端近傍におけ る体積増加が起こり,結果として全体排水量は増加して いる.その傾向はCase W03ではより促進されている.

上述した形状変更における体積分布移動および変化の結果は、Fig.8に示す長手方向断面積分布、すなわち CP曲線でも顕著な差となって示されている。今回の最 適解の一般的傾向は総じて肩落ちであり、Case W02 およびW03においては船尾端直前の体積が目立って増 加していることが分かる.この形状変更傾向は、後 に示すように伴流分布を減速させる方向に働いてい る.一方、初期船型値比較の浸水表面積はW02で約 0.8%増加、またW03で約1.6%増加となっており、こ



Fig. 11 Wake distributions at propeller section for the original and optimized hull forms

れはW02において船尾水面近傍の張り出しが初期船型 よりも大きく、よってガース長が増し、W03において はその傾向がさらに促進された結果である.

Fig.9 および Fig.10 には Case W00 および W03 の形 状を初期船型と比較して示す. Case W02の結果は Case W00とW03のほぼ中間的な形状であった. Case W03 の注目すべき点は、船尾近傍フレームラインの変曲点 が2つとなり、ガース中央近傍で外に張り出す形状が得 られたことである.船尾船底部近傍のフレームライン 形状は概ねV型に移行し、これにより初期船型の船尾 近傍ビルジ部で見られる Pressure pocket(低圧領域)が 平坦化され、それにともなって近傍流場の三次元剥離が 減少したこと確認した.また船体後半船底部の体積が より圧力の高い船尾水線面近傍に移動し、これらの変 形によって初期船型のものより圧力抵抗が減少してい る. またガース中央近傍の張り出しは、プロペラ流入速 度を減速させる方向で行われた形状変更であると考え られる. Case W03の変形傾向は従来の粘性抵抗最小化 計算では見られなかったものであり、今後実験によっ て確認する必要がある重要な知見であると思われる.

最後に,最適解の伴流分布の傾向を検討する.Fig.11 には初期船型および最適化船型の伴流分布を,Fig.12 には断面内流速ベクトルを示す.図中,今回想定した

-271-



Fig. 12 V-W vectors at propeller section for the original and modified hull forms

プロペラ面も示してある.図に示されているように, 最適化船型の船尾縦渦は初期船型のものに比べ総じて 減少している. W02および W03の渦が W00よりも順 に強められているが、これは前述した粘性抵抗の傾向 と合致するものである.一方,最適化船型全てのプロ ペラ面伴流は初期船型のものより減速されており、ま た粘性抵抗最小化のW00よりもW02がより減速され, それがW03においてはさらに減速されていることが分 かる.これは既に述べたとおり, αが大きいほど目的 関数減少の際のWnを減少させる比重が増すためであ り,前述したWnの傾向と合致している.これらの流 場の変更傾向は,従来の設計法で使用されるものとほ ぼ合致するだけでなく、今回用いた目的関数および設 計定数の設定において当初ねらった傾向と一致してお り、本手法および計算結果の妥当性を示すものと考え ることができる.

3.4 従来の船型設計法の観点に基づく解の評価

従来の船型設計においては,抵抗最小化と推進効率 最大化は必ずしも同一ステージでの扱いとなっておら ず,前者が先行し,それにより与えられた船尾流場で 作動する推進系の最適化が実行されるのが一般的であ る.本研究では両者を結合するキーとして伴流を採用 することで伝達馬力を目標関数に設定し,上記を同時 に取り扱うことが可能となった.

従来法では抵抗最小化が第一目標となっていること から、それに関連して伴流係数の値は小さくなり、設 計される推進系は低回転大直径化される傾向となるこ とは否めない.勿論、推進器との適当なクリアランス を保持するために船尾形状の変形は考慮され、これが 抵抗最小化にとってペナルティーとなるため、これを 考慮することで従来法でも伝達馬力最小化はある意味 で達成されているとも考えられる.しかしながら、こ れは二次的であり、本研究の場合はむしろ主船体形状 との直接的な関連を考慮しており、同一抵抗でもより 伴流係数の高い船型を探索する過程を採っている.そ の結果、従来法と比べて推進器直径はより小さくでき ることが期待でき、建造コスト面でも有利な船型の存 在を示唆しているものと言える.

そのようにして得られた船型は計算結果に見られる ように,船尾船底部近傍フレームラインに変曲点が2 つあることが特徴となっている.これは従来の設計法 の発想では到達し得ない解であり,伴流分布を制御す る上で有効な船型タイプとして期待できる.

一方,本研究の設計では伴流分布において,推進器 トップ部にキャビテーション・起振力上,不利な高伴 流域を発生させる可能性があり,実用の観点からはハ イスキュー・起振力軽減装置の採用の検討によりある 程度は対処可能であるが,最適化過程での適当な制約 条件の設定が今後の課題と言える.

4. 結言

本研究では、RaNS法とSQPに基づく非線形計画法 を結合し、近代タンカー船型の船尾形状最適化を行 う計算手法の開発を行っており、本報では特に一定計 画速度におけるDHP最小化を目指した船尾形状最適 化について検討した.RaNS法には近年著者らがタン カー船型を対象として開発してきた方法を用い、今回 乱流モデルを改良することによってプロペラ面内の平 均流速場計算精度を向上させた.本報では、まず一定 計画速度DHP最小化が抵抗と伴流値を変数とした単 一目的関数の問題に帰着することを示し、実際の設計 経験値を用いて目的関数の経験定数を決定した.そし て粘性抵抗最小化およびDHP最小化計算を行い、経 験定数が解に与える影響を調査するとともに今回提案 した目的関数の妥当性を検討した.

解として得られた船型形状および粘性流場を検討した結果,DHP最小化においては、粘性抵抗だけでなく伴流情報も包括的に考慮した目的関数の採用が有効であることが確認された.また本報で提案した目的関数は、粘性抵抗および伴流係数の増減に関する相互関

-272-

係を表現する上で妥当なものあり,今回設定した条件 においては有効に機能することを確認した.さらに実 際の設計経験値に最も近い値を用いたDHP最小化で は,従来の粘性抵抗最小化計算では見られなかった形 状が得られ,これは今後実験によって確認する必要が ある重要な知見であると思われる.一方,伴流分布一 様化や推力減少率改善の見地で有効な制約条件の検 討・導入は今後の課題である.

参考文献

- Tahara, Y., Himeno, Y., and Tsukahara, T.: An Application of Computational Fluid Dynamics to Tanker Hull Form Optimization Problem, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 515-531
- 2) Hino, T., Kodama, Y., and Hirata, N.: Hydrodynamic Shape Optimization of Ship Hull Forms Using CFD, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 533-541
- 日本造船研究協会:第229研究部会「数値流体力 学による最適船型設計法の研究」報告書(第3年 度),1999
- 田原裕介,齋藤泰夫,姫野洋司:CFDによるタン カー船型の船尾形状最適化-第1報:粘性抵抗最小 化-,関西造船協会誌,第231号,1999, pp.29-36
- 5) 浜崎準一,姫野洋司,田原裕介:非線形計画法によ る船型最適化の試み(第4報)-伴流と粘性抵抗 を考慮した船体後半部形状の改良-,関西造船協会 誌,第226号,1996,pp.15-21
- Baldwin, B.S. and Lomax, H.: Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, 1978, pp. 1-8
- 7)日本造船研究協会:第222研究部会「大型肥大船船 尾流場推定法の高度化」報告書(第3年度),1996
- Tahara, Y. and Himeno Y.: Application of Isotropic and Anisotropic Turbulence Models to Ship Flow Computation, J. Kansai Society of Naval Architects, Japan, No. 225, 1996, pp. 75-91

論

討

[討論] (日立造船(株))瀬崎 良明 DHPの評価には1-tの影響も入れるべきと思います がいかがでしょうか. [回答]

ご指摘のとおり、1-t変化を考慮することは有益で す.一方、実際に $W_n=1-w_n$ の減少に伴って1-tが減少 する場合、その減少量のオーダーは W_n の減少量のオー ダー10⁻¹程度に対してより小さいと思われ、したがっ て本研究の方法によっても妥当な結果が得られている と考えています.

[討論] (NKKエンジニアリング研究所)岡本 幸彦 今回の計算は低レイノルズ数の計算と思われますが, 船型最適化に関するレイノルズ数の影響について調査 されていればお教えください.

[回答]

ご指摘のとおり、今回の計算はモデルスケールで行っ ています.現在フルスケール CFDの検討を行ってお り、その研究においてご質問のレイノルズ数影響の調 査を詳細に行う予定です.有益な結果が得られ次第ご 報告いたします.

CFDによるタンカー船型の船尾形状最適化(第3報) - 自航シミュレータを用いた伝達馬力最小化 - *1

正 会 員 田原 裕介 *², 正 会 員 西田 隆司 *³ 正 会 員 安東 潤 *⁴, 正 会 員 姫野 洋司 *²

CFD-Aided Optimization of Tanker Stern Form (3rd Report) - Minimization of Delivered Horse Power Using Self-Propulsion Simulator -

By Yusuke TAHARA (Member), Ryuji NISHIDA (Member) Jun ANDO (Member) and Yoji HIMENO (Member)

This paper presents optimization of tanker stern form based on viscous flow information obtained from Computational Fluid Dynamics. In this 3rd report, main focus is placed on minimization of delivered horse power in conjunction with application of self-propulsion simulator, which is based on extension of a Reynolds-averaged Navier-Stokes (RaNS) equation solver including propeller effects evaluated by the infinite-number propeller theory. The overall numerical method follows that of earlier work (1st and 2nd reports), i.e., coupling of RaNS equation solver and successive quadratic programming. Constraints considered in the present study are basically same as those used in earlier work, i.e., geometrical requirements of optimized hull for displacement, profile, maximum beam and depth, and stern frame lines to secure enough space for engine room. In the following, details of the numerical method are described, and the discussion of the results includes identification of salient differences of forms and integral parameters between the original and optimized hull forms. In conclusion, the present method appeared to successfully optimize the given tanker stern form, and the modification trends automatically demonstrated in the present work agree well with those commonly in use in traditional tanker hull form design.

Keywords : Tanker Hull Form Optimization, Reynolds-averaged Navier-Stokes Equations, Successive Quadratic Programming, Minimization of Delivered Horse Power, Self Propulsion Simulator

1. 緒言

推進効率向上を目指した船尾形状改良/最適化を行 う場合,抵抗最小化と推進効率最大化,さらにプロペ ラ影響を考慮したプロペラ流入面の伴流分布最適化な どを同時に考慮した上で,最適形状を求める必要があ る.従来の設計法では抵抗最小化が先行し,それに より与えられた船尾流場で作動する推進系の最適化

- *3 石川島播磨重工業(株)(研究当時 大阪府立大学大学院 工学研究科)
- *4 九州大学大学院工学研究院

が実行されるのが一般的である.本研究では,それ らを同ステージで達成する手法の開発を目的として, 計算流体力学(Computational Fluid Dynamics: CFD) と非線型計画法の結合に基づく,タンカー船型の船尾 形状最適化計算法の検討を行っている^{1),2)}.

CFDと非線型計画法を結合し,船型形状最適化を 試みた研究には、当初は非粘性自由表面流場計算法 や境界層方程式法などの比較的簡便な流場計算法が 応用され,近年はレイノルズ平均ナビエ・ストークス 方程式法(Reynolds-averaged Navier Stokes Equation Method: RaNS法)の適用も始まっており¹⁾⁻⁹⁾,著者ら の研究は後者に属するものである。著者らは前報^{1),2)} までの研究において,粘性抵抗最小化および簡便な伝

^{*1} 平成12年5月25日関西造船協会春季講演会において講演,原稿受付 平成12年6月9日

^{*2} 大阪府立大学大学院工学研究科



Fig. 1 Definition sketch of coordinate system.

達馬力(Delivered Horse Power: DHP) 推定式を用い たDHP最小化を検討した.その結果,本研究で開発 する計算手法のカーネル部は有効に機能しており,ま た解が示す傾向は従来型設計法と合致したものである ことに加え,特にDHP最小化に関しては,従来法で は到達しがたい船型変更傾向を提案できる可能性があ ることも確認した.

一方,著者らが行ったDHP最小化²⁾においては,推 力減少率やプロペラ吸引・回転影響は厳密に考慮せず, また船型変更の度に最適プロペラを設計すると仮定し, 船型に応じた経験定数を導入することによって,最小 化目的関数である簡便なDHP推定式を導いた.他方, 特定自航状態において上述したプロペラ影響を重視し, かつDHP最小化を試みる場合には,自航状態を再現 するRaNS法,すなわち自航シミュレータの応用が有 効である.自航シミュレータに関しては,Sternら¹⁰⁾ やそれに類似した方法による研究例¹¹⁾⁻¹³⁾が報告され ているが,船尾形状最適化に応用した研究はこれまで に報告されていない.

本報の主眼は,自航シミュレータと逐次二次計画 法(Successive Quadratic Programming: SQP法)の結 合による,DHP最小化を目指したタンカー船型の船 尾形状最適化である.自航シミュレータは,近年著者 らがタンカー船型を対象として開発してきたRaNS法 ^{1),2),6)-8)}に,無限翼数プロペラ理論に基づくプロペラ 影響・性能計算法^{14),15)}を搭載し,本研究において新規 に開発したものである.著者らは,前報^{1),2)}までの計 算手法をさらに拡張し,自航状態における船尾粘性流 場に及ぼすプロペラ影響を考慮するとともに,プロペ ラ性能計算によって得られるDHPを目的関数とした DHP最小化を試みた.本論文では,まず今回開発し た自航シミュレータの精度を確認した結果について述 べ,つづいてDHP最小化を目指したタンカー船型の 船尾形状最適化の計算結果について議論する.



Fig. 2 Computational procedure of the present self-propulsion simulator.

2. 自航シミュレータの概要

2.1 計算手法の概要

流場の基礎方程式は、デカルト座標系(Fig.1)で記述 された、3次元非定常、非圧縮流体に関するRaNS方程 式である.本研究で用いたRaNS法および乱流モデル、 また格子トポロジー(C-O型)は、基本的に前報^{1),2)}の ものと同一である.今回拡張した点は、左右舷で非対 称なプロペラ影響を考慮するために両舷計算としたこ と、計算格子を改良したこと、さらにプロペラ影響・ 性能計算法を搭載したことが挙げられる.DHPやプ ロペラ性能、さらにプロペラ影響の計算には、無限翼 数プロペラ理論に基づく手法^{14),15)}を用いた.その手 法に関しては、SR222研究部会¹³⁾やSR229研究部会⁸⁾ において、RaNS法への搭載例が詳細に検討されてい る.プロペラの形状や位置などについては、SR229研 究部会で標準的とされたものと同一にした.なお今回 の計算においては、自由表面影響は考慮していない.

つづいて自航計算法の概要について述べる.手順の 概要をFig.2に示す.計算の初期流場には, 曳航状態の



Fig. 3 Cross-sectional view of computational grid at propeller section.

計算結果を使用し、つぎのような手順で自航状態の解 を求める:(1)RaNS法の解(U,V,W)よりプロペラディ スク扇形要素の計算代表点上の値を求め、さらにプロ ペラ誘導速度を除いて有効伴流分布を求める; (2)プロ ペラ性能計算ルーチンをroot finderルーチン(二分法 を使用)より起動し、前段階で求めた流場に基づく粘性 抵抗Rと釣り合う推力Tをもたらすプロペラ回転数n_n を求める;(3)前段階において求めたnpを用いてプロ ペラ性能計算を行い、プロペラ吸引・回転作用に等価 なRaNS方程式における体積力(body force)を計算す る;(4)前段階で求めた体積力を用いてRaNS方程式を 解く;(5)前段階の流場に基づく粘性抵抗Rが許容誤差 (0.1%)以内に収束した場合は自航状態に至ったとして 計算を終了し、それ以外はステップ(1)に戻る、今回の 計算においては、自航計算を開始して無次元時間6.0 までには自航状態に達することを確認した.

一方,計算格子の生成にはポアソン方程式法を用いているが、今回は計算空間の座標 ξ , η , ζ 方向(それぞれ長手方法、物体法線方向、ガース方向)の制御関数を改良し、スターンバルブ近傍の格子直交性を向上するとともに、将来実船レベル・レイノルズ数(Rn)の計算法へ拡張する準備として、全体的な格子品質を向上させる技術を導入している.なお壁面近傍の最小格子間隔は $y^+ \approx 1$ となるように設定した.後述する設定Rn以上の $Rn=2 \times 10^7$ で試計算を行った結果、曳航状態の粘性抵抗および流場の傾向は妥当なものであることに加え、計算安定性も明らかに向上していることを確認した.実船レベルRnに対応した計算に関しては、将来報告する予定である.

2.2 計算結果の評価

本研究では、まず今回開発した自航シミュレータの 精度確認を行った.供試船型はSR221B船型とし、設定 $Rn(=U_{\infty}L/\nu, L, \nu$ はそれぞれ船速,船長,流体の動粘 性係数)は、実験状態⁸⁾に適合させて $Rn=8 \times 10^{6}$ とし た.計算格子に関しては、船尾近傍の流場解像度向上を



vectors at propeller section (SR221B for towing condition).

考慮し,前報^{1),2)}および後述する最適化計算で定義す る密格子よりさらに格子数を増やした計算格子(200× 40×60=480,000,それぞれ ξ , η , ζ 方向)を用いた.プロ ペラ断面における計算格子の状態をFig.3に示す.以下 の議論において,計算結果は設定一様流速 U_{∞} ,船長Lで無次元化して示す.なおX=0,1はそれぞれFP,APに 対応することに注意されたい.また,以下で述べる計算 および実験の結果は,ともに舵なし状態のものである.

曳航状態のプロペラ断面における伴流分布(U)と断面内流速ベクトル(V-W)をFig.4に,さらに自航状態のプロペラ直前と直後の断面における計算結果をFig.5とFig.6に示す.曳航状態においては,肥大船の船尾流場に特有の強い縦渦と,それにともなうフック型の伴流分布が見られ,それらの大まかな傾向は実験値¹⁶⁾のものと良く一致している.

自航状態のプロペラ直前断面では,船尾縦渦の中心 は船体中心面方向に引付けられ,また伴流はプロペラ ディスクの直前で全体的に加速されている.また曳航 状態においては顕著であった伴流分布のフック形状は,



Fig. 5 Axial wake distributions and V-Wvectors at propeller inlet section (SR221B for self-propulsion condition).

自航状態では不鮮明になる.これらはプロペラ吸引作 用による縮流効果によるものである.自航状態では, 伴流分布は左右非対称となり,これは後述するプロペ ラ面内推力分布の左右非対称性とも関連する効果であ る.なおV-Wベクトルの様子も左右非対称となってい るが,この断面においては回転方向に誘起されるプロ ペラ誘導速度の影響は見られない.

一方,自航状態のプロペラ直後断面では,プロペラ の旋廻影響を強く受けた流場の様子が確認でき,プロ ペラ軸に関して同心円状の伴流等値線や,プロペラ回 転方向に誘起された強いV-Wベクトルなど,実験値 ¹⁶⁾に見られる多くの特徴が正確に再現されている.ま た,その回転中心が若干左舷によっている点も実験値 と類似している.

自航状態のプロペラ面内推力分布および有効伴流分 布をFig.7に示す.有効伴流分布は,RaNS法で求まる 軸方向のプロペラ流入速度分布より,プロペラ誘導速 度分布を差引いたものである.前述したように,有効 伴流分布は左右で非対称となっている.両舷で船体中

-277-



Fig. 6 Axial wake distributions and V-Wvectors at propeller exit section (SR221B for self-propulsion condition).

心面(Y=0)に対称な2点の値を比較すると,有効伴流 値が低いほど推力値はより大きくなっており,これは 今回用いたプロペラ理論に合致する傾向である.

推力分布が左右非対称となったことついては,以下 の理由が考えられる.今回行ったプロペラ計算におい ては,プロペラ流入面における二次流れ成分を考慮し ている.特定方向(ここでは右回転)に回転するプロペ ラからみた二次流れ成分は,事実上左右舷で異なるた めに,左右舷で異なった推力分布が生じることになる. この効果は痩型船型では顕著でないと思われるが,船 尾流場の二次流れ成分が大きい肥大船の場合は,比較 的明瞭に現れたと考えられる.

曳航状態と自航状態の船体表面圧力分布の比較図を Fig.8に示す.図中,Cptowは曳航状態を,またCpself は自航状態を示す.さらにそれらの差Cpself-Cptowも 示しており,これは推力減少係数1-tを決定する支配的 要素となる.自航状態ではプロペラディスクの直前で プロペラ吸引作用の影響が明瞭に見られ,曳航状態よ り低圧領域が拡大している.またプロペラ影響は比較



Fig. 7 Axial body force (a) and effective wake (b) distributions on propeller disk (SR221B for self-propulsion condition).

的船尾端近傍に限定された領域で顕著である.比較で きる実験値は左舷のみの結果¹⁶⁾であるが,その一般的 傾向は正確に再現されていることが確認できる.伴流 分布において指摘した点と同様,自航状態における表 面圧力場にも左右非対称性が認められ,これは本報の ものに類似した計算手法を,他船型に適用した場合に も指摘されている¹¹⁾.

最後に表面積分値の結果を検討する.まず曳航状 態の1+Kおよび公称伴流係数1- w_n の結果をFig.9に示 す.若干の差は認められるものの,計算値と実験値は 良く一致している.つぎに自航要素の実験値⁸⁾との比 較をFig.10に示す.1- w_t は推力一致法で求めた有効伴 流係数, η_r はプロペラ効率比,1-tは推力減少係数であ る.1- w_t は実験値のばらつき範囲内の値を示し,また η_r はほぼ1.0となり実験値と同等である.一方1-tは今 回の計算においてやや過小評価されているが,その原 因については,RaNS法においてプロペラボスの影響 が考慮されていないことなどが指摘でき,今後改善を



Fig. 8 Comparison of surface pressure distributions between towing and self-propulsion conditions (SR221B).

検討すべき課題である.なお1-tに関連する自航状態の 抵抗増加は,圧力抵抗の増加分が支配的であり,摩擦 抵抗の増加は1%程度であった.

以上の結果,今回の計算において,曳航/自航状態 の流場や表面積分値の傾向に関し,実験値に見られる 重要な傾向は,ほぼ正確に再現されていることを確認 した.1-tに関しては定量的精度に関し課題が残るが, 本報で開発した自航シミュレータは,後述する船尾形 状最適化計算に適用する範囲では,満足できる計算精 度を有するものと考えられる.

3. 伝達馬力最小化計算

3.1 計算手法の概要

ここでは、上述した自航シミュレータを応用し、 DHP最小化計算を行った結果について述べる.本研 究で採用した6パラメータ形状変更関数,SQP型最適 化アルゴリズム,排水量(DISP.)制約条件(DISP.は 初期船型のもの以上),プロファイル一定条件,主機配 置要件に基づくS.S.7/8フレームライン制約条件,最 大喫水および最大半幅一定制約条件,形状変更領域 (S.S.4~船体後端),さらに設定Rn=1.9×10⁶は,前報

-278-


Fig. 9 Comparison of 1+K and $1-w_n$ between computations and experiments (SR221B for towing condition).

^{1),2)}で使用したものと同一である.加えて前報と同様,最適化計算は粗計算格子(90×40×60=216,000) で行い,得られた船型に基づく密計算格子(170×40×60=408,000)を生成し,後者の結果を用いて流場情報の解析を行った.今回の最適化初期船型は,前報でも採用したSR221X船型(SR229研究部会⁸⁾においてはSR221B-L2船型と定義)である.またプロペラ位置や直径などの幾何情報については,SR229研究部会で標準的とされたものを固定して使用しており,前報²⁾で仮定した形状変更ごとの最適プロペラ設計は行っていない.なお今回の計算では,前報²⁾と同様,自由表面や舵の影響を考慮していない.

以下の議論において,初期船型の解をOriginal, 粘性抵抗最小化の解をVR-Min.(Viscous Resistance Minimized)および DHP最小化の解を DHP-Min.(DHPMinimized)と定義する.今回行った DHP最小化計算 では,最適化サイクル8回において計算が終了した.ま たS.S.7/8におけるフレームライン条件や,その他の幾 何学的制約条件は,数値誤差の範囲で全て満足されてい ることを確認した.なお VR-Min.の解は,前報²⁾で得 られた粘性抵抗最小化の解(Case W00)と同一である.

3.2 計算結果の評価

排水量 DISP., 浸水表面積 WSA, 粘性圧力抵抗 ($Rv=Cvp\cdotWSA$), そして DHPの比較を Fig.11に示 す. VR-Min.の DHPは Original のものに比べ約6%減 少している. これは, DHP低減のためには Rvの低減 の影響も大きいからである. 一方 DHP-Min.の DHP は VR-Min.のものよりさらに約1%減小し, Fig.11に 示す計算結果のなかでは最小となった. この傾向は初 期の目的と合致しており, 本最適化手法が有効に機能 していることを示すものと考えられる. Rvは VR-Min. が最小で, DHP-Min. ではそれより約1%増加するが,





これはDHP-Min.で見られる1- w_t 低減と関連する効 果であり、その詳細は後述する. Rv最小化が必ずし もDHP最小化とならないという、従来の設計法で用 いられる概念と合致する結果である.なおDISPや WSAに関しては、VR-Min.において数値誤差の範囲 でOriginalと同等であるのに対し、DHP-Min.ではそ れより若干増加する傾向がある.一方、Rvの増減に関 しては、前報^{1),2)}において指摘したように、粘性圧力 抵抗の影響が支配的であることを確認した.

つぎに自航要素 $(1+K,1-w_t,\eta_r,1-t)$ の比較をFig.12 に示す. 1+Kに関しては上述のRvと同傾向であり, η_r に関してはOriginalとほぼ同等(定量的数値は約1.0) である. 一方,推力一致法で求めた有効伴流係数1- w_t はVR-Min.においてOriginalより増加するのに対し, DHP-Min.においては減少する傾向が見られ,これは





DHP-Min.においてよりプロペラ効率が上昇したこと を示している.さらに1-tはVR-Min.においてOriginal より低下(悪化)するのに対し,DHP-Min.においては Originalと同レベルまで回復した.これらの自航要素 の傾向は,つぎに述べる最適解の形状の差と密接に関 連していると考えられる.

長手方向断面積分布, すなわち CP曲線をFig.13に, また長手方向断面積分布の最適化船型と初期船型 との差をFig.14に示す. さらに VR-Min. と DHP.-Min. の形状をそれぞれ Fig.15 と Fig.16に示す. VR-Min. と DHP.-Min. ともに体積分布が後方へ移動する点につい ては同じであるが, VR-Min. ではSS.1・1/2(X=0.85) 近傍より船首側で見られるU型→V型フレームライン 変更が顕著であるのに対し, DHP.-Min. では比較的U



Fig. 12 Comparison of computed 1+K, $1-w_t$, η_r , and 1-t between the original (SR221X) and optimized hull forms (VR-Min. and DHP-Min. as Rv and DHP minimized hulls, respectively).



Fig. 13 Distribution of cross-sectional area (CP curve) for the original (SR221X) and optimized hull forms (VR-Min. and DHP-Min. as Rv and DHP minimized hulls, respectively).

型を保つ傾向がある.結果として、VR-Min.のCP曲 線には明らかな肩落ち傾向が認められる.U型→V型 フレームライン変更が粘性抵抗に与える影響は、船尾 ビルジ近傍で起こる境界層の3次元剥離と、それにと もなう船尾縦渦の生成に関連があり、詳細は前報^{1),2)} を参照されたい.一般にU型→V型フレームライン変 更が進行すると粘性抵抗は低下するが w_t が低下し、プ ロペラ効率は低下する.今回DHP.-Min.で見られた傾 向は、粘性抵抗を低下させつつもある程度でその傾向 を抑制し、 w_t の増加も図ったものと考えられる.

他方,体積分布の後方移動が極端に進行すると1-t の低下をもたらし,結果的にはVR-Min.においてそ の傾向が見られる.DHP-Min.の体積分布の後方移動 は,VR-Min.より規模が小さく,また特にプロペラ上



Fig. 14 Differences in cross-sectional area between the original (SR221X) and optimized hull forms (VR-Min. and DHP-Min. as Rv and DHP minimized hulls, respectively).

部近傍の直前においてWaterlineの正接角がより小さ くなっている.これらの変更傾向は1-tの低下を抑制す る観点で妥当なものである.DHP.-Min.において見ら れた傾向は,体積分布の後方移動を適度に行い,1-w_t を減少させながら,同時に1-tの悪化を避けるという, 従来の設計法と合致する操作を自動的に再現したもの と考えられる.

3.3 前報²⁾の解との比較

ここでは、前報²⁾で得たDHP最小化の解と、今回の 解との比較を行う.前報のDHP最小化において使用し た DHP 推定式は、 DHP^{α_0/α_1} $\propto RvW_n^{\alpha_2/\alpha_1} = RvW_n^{\alpha_2}$ で表される.ここで $W_n=1-w_n$ は公称伴流係数,Rvは粘 性抵抗,そして $(\alpha_0, \alpha_1, \alpha_2)$ は設計経験値であり,例とし てタンカー船型の場合には(0.6661,1.1670,0.3322)と設 定できる.同式を導く過程で導入した仮定は,形状変更 の際に航行速度, 1-t, そして η_r は一定, 1- w_e (有効伴流 係数) $\propto 1-w_n(=W_n)$ および前進率 $J \propto W_n/n_p D(n_p:\mathcal{T})$ ロペラ回転数,D:プロペラ直径), さらにエンジンはト ルクー定制御としたことである.加えて諸量の局所線 形性を仮定し、形状変更の度に最適プロペラを設計す るとしている.本報のDHP最小化においては, 航行 速度は一定としたことを除き、上述した仮定は使用し ていない.一方、今回は一定プロペラを使用した自航 状態における DHP 最小化を目的としたため、実際の 設計で行われる形状変更ごとの最適プロペラ設計を考 慮するには至っていない. それらの設定条件が異なる ために, 今回と前報の解を直接比較することは困難で あるが、最適解の一般的傾向として重要と思われる点



Fig. 15 Comparison of buttock and water lines, and body plan between the original (SR221X) and optimized hull forms (VR-Min. as the modified).

に着目する.

まず排水量と浸水表面積の傾向は、既述した Original, VR-Min.,そしてDHP-Min.の間で前報²⁾と同様な傾向 が見られる.例えば排水量は、VR-Min.においては制 約条件で制限されるまで減少するのに対し, DHP-Min. ではOriginalよりむしろ増加させる傾向があり、また 体積分布はともに後方へ移動している。また、粘性 抵抗Rvの順位とDHPの順位の傾向も前報と同一で, DHP-Min.の Rvは VR-Min. より大きいが DHPはより 小さい、一方、形状変更の傾向に関しては、船尾近傍 のフレームラインに明確な2つの変曲点が現れた前報 の解に比べ、今回の解は比較的初期船型の形状を維持 したものとなっている.これは特定のプロペラを使用 した場合について1-t悪化を抑制するため、体積分布の 後方移動が抑制されたこと,また前回と同様今回も主 機配置に基づくフレームライン制約条件を課している が、今回その制約がより強く影響を与える結果となっ たこと、などが考えられる、本研究で使用する非線型



Fig. 16 Comparison of buttock and water lines, and body plan between the original (SR221X) and optimized hull forms (DHP-Min. as the modified).

計画法のアルゴリズムは、制約条件や形状変更関数の 特性が、解の挙動に影響を及ぼすことがあることも報 告されており^{1),2),4)-6),8),9)},それらについては今後よ り詳細に再検討する余地がある.

一方,今回の方法も前報²⁾と同様,プロペラ流入面の 伴流分布自体の要素を目的関数に考慮しておらず,最 終的な伴流分布に関し,推進器トップ部にキャビテー ション・起振力上,不利な高伴流域を発生させる可能 性がある.実用の観点からはハイスキュー・起振力軽 減装置の採用の検討によりある程度は対処可能である が,最適化過程において適当な制約条件を設定するこ とが望ましい.前報に引続き,それらは今後の課題と して残されている.

4. 結言

本研究では、自航シミュレータとSQPに基づく非線 型計画法を結合し、DHP最小化を目指したタンカー 船型の船尾形状最適化を行った.自航状態における船 尾粘性流場に及ぼすプロペラ影響を考慮するとともに、 プロペラ性能計算によって得られる*DHP*を目的関数 とした*DHP*最小化を試みた.本研究では,まず今回 開発した自航シミュレータをSR221B船型に適用し, 計算結果を実験値^{8),16)}と比較することによって,計算 手法の精度を確認した.つづいて本自航シミュレータ を応用した*DHP*最小化計算を行い,前報の解²⁾との比 較も含め,計算結果の妥当性を検討した.それらの結 果を以下に要約する.

(1)本自航シミュレータは,表面圧力場やプロペラ後 方流場に関し,プロペラ吸引・旋廻影響による重要な 流場傾向を,ほぼ正確に再現できることを確認した.

(2)本自航シミュレータの自航要素の計算精度に関 しては、1-tについてはやや過小評価する傾向が確認さ れるものの、 $1+K,1-w_n,1-w_t,\eta_r$ については満足できる 精度を有することを確認した.

(3)DHP最小化船型のDHPは明らかに初期船型の ものより小さく,また粘性抵抗最小化のものより小さく なった.これは本研究の初期の目的と合致した傾向で あり,本手法の妥当性を示す結果の一つと考えられる.

(4)DHP最小化船型の粘性抵抗は,粘性抵抗最小化のものに比較してより大きい反面,1-wtはより低く,また1-tはより大きくなることにより,結果的にDHPはより小さくなることが分かった.

(5)DHP最小化において見られる形状変更傾向は, 体積分布の後方移動を適度に行い,1-w_tを減少させな がら,同時に1-tの悪化を避けるという,従来の設計法 と合致する操作を自動的に再現したものであることを 確認した.

(6) 今回の DHP 最小化の解を, 簡便な DHP 推定式 を用いた前報²⁾の解と比較した結果, 粘性抵抗最小化 船型の解との相対的な関係に関し, 排水量と浸水表面 積の傾向や, 粘性抵抗と DHPの傾向に加え, 体積分 布の後方移動の傾向などについて, ほぼ同様な結果が 得られていることを確認した.

なお既述したとおり,自航シミュレータの1-tの予測 精度向上,またDHP最小化におけるより有効な形状 変更関数,最小化目的関数,さらに制約条件の検討は, 今後の課題として残されている.

参考文献

- 田原裕介,齋藤泰夫,姫野洋司:CFDによるタン カー船型の船尾形状最適化-第1報:粘性抵抗最小 化-,関西造船協会誌,第231号,1999,pp.29-36.
- 2) 田原裕介,齋藤泰夫,松山博志,姫野洋司:CFDによるタンカー船型の船尾形状最適化-第2報:伝達馬力最小化-,関西造船協会誌,第232号,1999,pp.9-17.
- 3) Larsson, L., Kim, K.J., Esping, B., and Holm, D.: Hydrodynamic Optimization Using

SHIPFLOW, Proc. The 5th International Symposium on Practical Design in Shipbuilding (PRADS'92), Newcastle, 1992.

- 4) 濱崎準一,姫野洋司,田原裕介:非線形計画法による船型最適化の試み(第3報)-粘性抵抗最小化を目指した船体後半部形状の改良-,関西造船協会誌, 第225号,1996,pp.1-6.
- 5) 濱崎準一,姫野洋司,田原裕介:非線形計画法によ る船型最適化の試み(第4報)-伴流と粘性抵抗を 考慮した船体後半部形状の改良-,関西造船協会誌, 第226号,1996,pp.15-21.
- 6) Tahara, Y., Himeno, Y., and Tsukahara, T.: An Application of Computational Fluid Dynamics to Tanker Hull Form Optimization Problem, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 515-531.
- 7) Hino, T., Kodama, Y., and Hirata, N.: Hydrodynamic Shape Optimization of Ship Hull Forms Using CFD, Proc. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, May 1998, pp. 533-541.
- 8) 日本造船研究協会:第229研究部会「数値流体力 学による最適船型設計法の研究」報告書(第3年 度),1999.
- 9) 試験水槽委員会シンポジウム:船型設計と流力最 適化問題,日本造船学会,1999.
- Stern , F., Kim, H.T., Patel, V.C., and Chen, H.C.: A Viscous-Flow Approach to the Computation of Propeller-Hull Interaction, Journal of Ship Research, Vol.32, 1988, pp. 246-262.
- 11) 日夏宗彦, 児玉良明, 藤沢純一, 安東潤: プロペラ影響を考慮した船体まわり流れの数値シミュレーション, 西部造船学会々報, 第88号, 1994, pp.1-12.
- 12) 日夏宗彦,日野孝則,児玉良明,藤沢純一,安東潤:自航 状態における舵付き船体周り流れの数値シミュレー ション,西部造船学会々報,第90号,1995,pp.1-9.
- 日本造船研究協会:第222研究部会「大型肥大船船 尾流場推定法の高度化」報告書(第3年度),1996.
- 14) 中武一明:自航推進性能の計算法,船型設計のための抵抗・推進理論シンポジウム,日本造船学会,1979,pp.239-257.
- 15) 安東潤:第7回推進性能委員会シンポジウム「コンピュータ時代の船型開発技術」,日本造船学会,1997,pp.59-90.
- 16) 日本造船研究協会:第229研究部会「数値流体力 学による最適船型設計法の研究」報告書(第2年 度),1998.

論

討

[討論] 田中一朗

感じで申し訳ないことですが,前刷Fig.1と2(本稿 におけるFig.9とFig.10に相当)に示されている W_n と W_t の差は結構大きいなと思いながら眺めています.タ ンカー船型だからでしょうか.自航シミュレータの精 度はチェックされていると思いますが,コンテナ船型 などではどうなりますか.

[回答]

ご指摘のW_nとW_tの差は約0.1程度であり,実験値 とも比較した限りでは妥当なレベルではないか,と考 えております.別研究においてコンテナ船型への適用 を検討しておりますが,その場合のW_nとW_tの差は, 実験値・計算値ともにより小さくなっています.

Wave Influences on Viscous Flow around a Ship in Steady Yaw Motion

Yusuke Tahara^{*}, Member

Summary

This paper concerns an investigation of wave influences on viscous flow around a ship in steady yaw motion, which is based on detailed analysis of numerical solutions of Reynolds-averaged Navier-Stokes (RaNS) equations. The present numerical method solves the unsteady RaNS and continuity equations for mean-velocity and pressure and wave fields with exact nonlinear kinematic and approximate dynamic free-surface boundary conditions. In the following, an overview is given of the present numerical approach, and results are presented for the Series 60 C_B =0.6 at yaw angle 10° for with/without-wave cases, including wherever possible comparison with experimental data. Then discussions are focused on identification of salient feature of wave-included effects on the mean-velocity and pressure fields, body surface pressure and frictional streamlines, and hydrodynamic forces acting on the hull. Lastly, some specific aspects of flow for wave breaking and wave-induced vortices are described.

1. Introduction

Although the steady yaw condition is an approximation to the ship maneuvering condition, it involves many features of interest that must accurately be evaluated in hull form design. The yaw condition is dominated by strong crossflow effects, including forebody and afterbody keel and bilge vortices. For ships operated at high Froude number (Fn), wave effects on flows and hydrodynamic forces are significant due to the larger wave amplitudes especially on pressure side and need to correctly be estimated $^{1),2)}$, on which, however, relatively little theoretical work has been done with the method comprehensively includes viscous and wave effects. Hence, the present study is motivated, i.e., it concerns an investigation on waveinduced effects on viscous flow around a ship in steady yaw motion.

Research on ship yaw condition had been initiated with experimental work, such as Ueno et. al. $^{3),4)}$

* Dept. Marine System Eng., Osaka Prefecture University

Received 30th May 1999

Read at the Autumn meeting 18, 19th Nov. 1999

for resistance tests for tanker, cargo and combatants. More recently, Nishio et al. ⁵⁾ and Okuno et al. ^{6),7)} carried out surface pressure and mean-flow measurements in wind tunnel and water channel, where important aspects of the three-dimensional flow separation were investigated. In a domestic Japan research project SR221 ⁸⁾, emphases were on tanker hulls with systematic investigation of hydrodynamic forces and flows in near wake region. However, the most detailed documentation on wave influences on viscous flow had not been completed until Longo et al. ^{1),2)}, who carried out extensive mean-flow and force measurements for the Series 60 C_B =0.6 yaw condition for various Fn.

Theoretical work on ship yaw condition had been set forth without wave effects, in which sideforce prediction and modeling of vortex layer were mainly focused. Fuwa ⁹⁾, Mizoguchi ¹⁰⁾, and Matsumura ¹¹⁾ applied slender-body approximation in conjunction with vortex-layer computation, then evaluated sideforce with practical accuracy; but, intrinsic limitation appeared to be clear because of lack of viscous effects in the theory. More recently, the present author ¹²⁾ and Izumi et al. ¹³⁾ developed Reynolds-averaged Navier-Stokes (RaNS) and Navier-Stokes (NS) equation methods, respectively. The results for NS/RaNS equation methods were shown to be promising and warrant further extension, although certain issues related to turbulence model were involved. In the abovementioned studies, however, wave effects on flow are not well focused.

On the other hand, considerable effort had been directed towards so-called Neumann-Kelvin problem to consider wave effects on hydrodynamic forces acting on yawed ship hull, in which both Havelockand Rankine-source approaches were used. The approaches had been developed by Nonaka 14), Hori et al. ¹⁵⁾, Song ¹⁶⁾, and Maruo et al. ¹⁷⁾. And more recently Nakatake et al. ¹⁸⁾, Xia et al. ¹⁹⁾, the present author $^{20)}$, Rosen et al. $^{21)}$, and Iwasaki et al. $^{22)}$ utilized Dawson-type panel method or its extended version with modified wave radiation condition. Those methods predicted wave field as well as hydrodynamic forces with considerable accuracy; however again, specific flow aspects for the yaw condition, those mentioned earlier, could not be resolved due to the lack of viscous effects in the theory.

It is clear that more comprehensive approach must be introduced to investigate the present problem. Method used in the present study is a RaNS equation solver to calculate viscous free-surface flow around a ship in steady yaw motion, i.e., both wavemaking and viscous effects are included in the theory. Recently, a few of such methods have been developed $^{23)-25)}$, including a method which has been modified and extended for the present study. In those work, focuses were placed on evaluation of capability of Computational Fluid Dynamics (CFD) for yaw condition, where numerics and accuracy of the CFD methods were mainly stated; but the detailed documentation on wave-induced effects on flow had been incomplete.

As mentioned earlier, the present study concerns an investigation of wave influences on viscous flow around a ship in steady yaw motion. The present numerical approach solves the unsteady RaNS and continuity equations for mean-velocity, pressure and wave fields with exact nonlinear kinematic and approximate dynamic free-surface boundary conditions. As the above-stated, a method precursory developed by the present author and others ²⁴ has been modified and extended for the present study so as to exactly follow experimental condition, and more detailed numerical and validation uncertainty analyses have been completed. In the following, an overview is given of the present numerical approach, and results are presented for the Series 60 C_B =0.6 at yaw angle 10° for with/without-wave cases, including wherever possible comparison with experimental data. Then emphases of discussion are placed on identification of salient feature of wave-included effects on the mean-velocity and pressure fields, body surface pressure and frictional streamlines, and hydrodynamic forces acting on the hull. Lastly, some specific aspects of flow for wave breaking and wave-induced vortices are described.

2. Computational method

The present numerical method is based on extension and modification of the method ²⁴⁾ or more basically the method ²⁶⁾ for the present problem. The major modifications on the method ²⁶⁾ are summarized as follows: (1) expansion of solution domain to include both port and starboard sides of the hull (See Fig.1), associated with modification of velocity and pressure boundary conditions; and (2) modification of numerical technique to regenerate free-surface conforming grid as the free-surface is updated, so that the initialgrid relocation method for zero-yaw case ²⁶⁾ is also used but independently utilized for both sides of the hull, that is separated by keel plane (see Fig.2). On the other hand, extensions made on the method ²⁴⁾ are: (1) inclusion of sinkage, heel, and trim condition so as to exactly follow experimental condition especially for hydrodynamic force measurements; and (2) refinement of time integral so that time level of velocity, pressure, and wave fields coincides, in order to extend the present method for dynamic simulation in future work. In addition, extensive grid studies are completed in the present study to evaluate numerical uncertainty, which is described later. In the following, an overview of the present method is given.

The present numerical method solves the unsteady RaNS and continuity equations for mean-velocity (U, V, W), piezometric pressure \hat{p} and eddy viscosity ν_t by using the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic freesurface boundary conditions, and a body/free-surface conforming grid. The equations are transformed from Cartesian coordinates (X, Y, Z) in the physical domain to numerically-generated, boundary-fitted, nonorthogonal, curvilinear coordinates (ξ^1, ξ^2, ξ^3) in the computational domain. A partial transformation is used, i.e., coordinates but not velocity components (U, V, W). The equations are solved using a regular grid, finite-analytic spatial and first-order backward difference temporal discretization, PISO-type pressure algorithm, and the method of lines. The computational grid is updated at each time step to conform to both the body and free surfaces.



Fig. 1 Definition sketch of coordinate system, solution domain, and boundaries.

As shown in Fig.1, consider a ship fixed in the uniform onset flow $\vec{U}_0 = (U_0 \cos\beta, U_0 \sin\beta, 0)$, where β is yaw angle. In this study, β is given so that the port and starboard sides are the pressure and suction sides, respectively. Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward. Fig.1 also shows the solution domain, which includes both port and starboard sides of the hull. Referring to Fig.1, the specified boundaries are the body surface S_h , the inlet plane S_i , the exit plane S_e , the outer boundary S_o , and the free-surface S_{ζ} which for without wave case becomes the symmetry waterplane S_w .

The boundary conditions for without wave case, i.e., double-model computation, are: on S_h , $(U,V,W) = \partial \hat{p} / \partial n = 0$ (where n is normal to the body); on S_i , freestream values are imposed, i.e., $U=U_0\cos\beta$, $V=U_0\sin\beta$, $W=\hat{p}=0$; on S_e , axial diffusion and pressure gradient are assumed negligible, i.e., $\partial^2(U, V, W)/\partial X^2 = \partial \hat{p}/\partial X = 0$; on S_w , $\partial(U, V, \hat{p})/\partial\xi^3 = W = 0$; and on S_o , $U = U_0 \cos\beta$, $V=U_0\sin\beta, W=\partial\hat{p}/\partial n=0$ (where n is normal to the surface). For with wave case, the boundary conditions are similar, except on S_{ζ} , where exact nonlinear kinematic and approximated dynamic free-surface conditions are applied on the exact free surface $\zeta(X, Y)$, which is determined as part of the solution; i.e., the dynamic conditions $\partial(U, V, W)/\partial\xi^3=0$ and $\hat{p}=\zeta/Fn^2$ $(Fn=U_0/\sqrt{gL}, L$ is ship length) are applied to velocity and pressure with ζ determined through the solution of the exact nonlinear kinematic condition using a Beam and Warming linear multi-step scheme with both explicit and implicit 4th-order artificial dissipation. The boundary conditions for ζ are: on S_i , $\zeta=0$; on S_o , $\partial\zeta/\partial n=0$; and on S_e , $\partial\zeta/\partial X=0$.

In addition, initial conditions are taken from freestream values for both with and without wave cases, and $\zeta=0$ for the former. See reference ²⁶⁾ for zero yaw case for more basic details of the present numerical method.



Fig. 2 Initial and free-surface conforming grids.

3. Experimental Data and Computational Conditions

The Series 60 $C_B=0.6$ ship model is selected in the present study, since extensive experimental data for the model in steady yaw motion are available ^{1),2)}. The computational conditions simulate the experiments, i.e., basically same or nearly equal values for Fn, Reynolds number $(Rn:=U_0L/\nu, \nu)$ is kinematic viscosity, sinkage, trim, heel angle, and β are used. Note that Fn=0 condition is to compute double-model flow, i.e., the free-surface is treated as a plane of symmetry.

A partial view of the body/free-surface conforming grid for Fn=0.316 and $\beta=10^{\circ}$ is shown in Fig.3. For zero and nonzero Fn cases, the inlet, exit, and outer boundaries are located at X=(-0.4, 2.0) and r=1. The first grid points off the body surface are located in the range $y^+=RnU_\tau y_n \leq 2$, where U_τ is wall-shear velocity and y_n distance normal to wall. The RaNS grids are prepared as follows: for zero Fn, $90 \times 40 \times 60=216,000, 90 \times 60 \times 60=324,000, 90 \times 80 \times 60=432,000$, and $108 \times 48 \times 72=373,248$ (longitudinal(ξ^1) × radial(ξ^2) × girthwise(ξ^3) directions, respectively); and for nonzero Fn case, $180 \times 40 \times 80=576,000$, $160 \times 60 \times 71=681,600$, $180 \times 60 \times 80=864,000$, and $180 \times 80 \times 80=1,152,000$; all of which are used to estimate numerical uncertainties.



Fig. 3 Overview of computational grid around a ship (Fn=0.316, $\beta = 10^{\circ}$, $Rn = 5.3 \times 10^{\circ}$).

The values of the time increment and underrelaxation factors for velocity and pressure are 0.01, 1, and 0.1 for zero Fn, and 0.01, 1, and 0.01 for nonzero Fn. The computations are carried out up to 40 and 80 non-dimensional times for zero and nonzero Fn, respectively. The computed hydrodynamic forces indicate small oscillations with minimal reduction in the residuals after 20 and 40 non-dimensional times for zero and nonzero Fn, respectively, that may be due to the unsteady nature of the flow involving threedimensional separation. The oscillation of the solutions is considered in the analysis of iteration uncertainty described below.

The verification procedure of the present results basically follows that in Stern et al. $^{27)}$, however, validation processes described in Coleman et al. $^{28)}$ are also needed. Order-of-accuracy studies on the finiteanalytic scheme were carried out by Stern et al. including the present author $^{27),29)}$, for both simple and practical geometries, in which it is stated that orderof-accuracies vary between about 1.5-2.5 depending on geometry and flow complexity. The numerical uncertainty analysis in the present study is based on that of Coleman et al. ²⁸⁾, which practically defines validation uncertainty U_V as $U_V^2 = U_D^2 + U_{SN}^2 + U_{SPD}^2$, where U_D and U_{SN} are experimental and numerical uncertainties, and U_{SPD} uncertainty based on sensitivity analysis of model coefficients. In the present study, U_{SPD} is ignored. U_D is provided by Longo et al. ^{1),2)} and U_{SN} can be decomposed into two parts, i.e., uncertainties related to grid and iterative convergence. The procedure to evaluate those is described in reference ²⁸⁾. The present results are summarized in Table 1.

Table 1	Summary c	a innerical	and ex	perimental	uncertainties
	•				

Condition		Fn=0			Fn=0.31	6
	U _{SN} (%)	U ₀ (%)	U _v (%)	U _{SN} (%)	U _D (%)	U _V (%)
Су	5.0	1.8	5.3	4.0	0.1	4.0
Gx 1	4.7	6.0	7.6	4.0	0.6	4.0
CMz	2.5	4.1	4.8	2.5	0.1	2.5
Wave Profiles	-	2.6	-	4.0	1.3	4.2
Wave Elevations	-	2.2	-	4.0	1.1	4.2
Mean Flow						
U (%U ₀)	4.0	1.5	4.3	4.0	1.5	4.3
V (%U₀)	3.0	1.5	3.4	3.0	1.5	3.4
W (%U ₀)	1.0	1.5	1.8	2.0	1.5	2.5
Ср	0.7	3.0	3.1	1.0	3.0	3.2
- · Not applicable	or neglig	vible				

In presentation of the results and discussions to follow, variables are non-dimensionalized using ship length L, free-stream velocity U_0 , and fluid density ρ . Note that X=0 and 1 correspond to FP and AP, respectively, and the port and starboard sides of the hull the pressure and suction sides, respectively.

4. Results

4.1 Wave field

First, the present results of wave field are discussed. The model and computational conditions are same as those of the experiment, i.e., the model is fixed up right; Fn=0.316, $Rn=5.3\times10^6$; and yaw angle $\beta=10^\circ$.

The wave profiles at the hull are compared with the experimental data in Fig.4. The results for both port and starboard sides demonstrate close agreement with the measurements with respect to wave amplitude, shape, and phase; where the agreement on starboard side is remarkable, i.e., the inviscid-flow methods generally overestimate the wave amplitude due to the lack of viscous effects. Also, local region wave field $(|Y| \leq 0.15)$ is well predicted as shown in Fig.5. Issues are seen in global region wave field, i.e., the amplitudes

160

are underpredicted and details are missing, which may be due to the insufficient grid resolution in the region. Although the issues are involved, the wave field is successfully predicted in the region on which main focus of the present study is placed.



Fig. 4 Comparison of wave profiles on hull $(Fn=0.316, Rn = 5.3 \times 10^6)$.



Fig. 5 Comparison of wave elevation contours $(Fn=0.316, Rn = 5.3 \times 10^6)$.

4.2 Mean-velocity and pressure fields

Next, wave influences on mean-velocity and pressure fields are discussed. As is the case for wave fields, the model condition is fixed upright and computational conditions are Fn=0.316 and $Rn=5.3 \times 10^6$ for high Fn; and Fn=0 (double-model computation) and $Rn=2.7 \times 10^6$ for low Fn; and yaw angle $\beta=10^\circ$ for both Fn; all of which basically follow the experimental condition. Note that experimental Fn for low Fn is 0.160; however, the wave effects are negligible except near the bow so that double model simulation can be applied.



Fig. 6 Comparison of computed axial-velocity U contours (upright condition).

Results for the mean-velocity and pressure fields are presented in Figs.6 through 9, i.e., contours of axial velocity U, piezometric pressure Cp (=2 \hat{p}), axial vorticity ω_x , and crossplane flow vectors V-W, respectively, for several representative stations, i.e., X=(0.1, 0.2, 0.2)0.6, 0.9). Those figures include both Fn=0 and 0.316 results, in which comparison between the two enables to identify the wave-induced effects on flows. Reader may refer experimental data at the same stations in Longo et al. $^{1),2)}$. For both Fn, the overall agreement between the present results and experimental data is satisfactory, such that important flow features dominated by strong crossflow effects and four major vortex system, i.e., forebody and afterbody bilge and keel vortices, are successfully simulated in the present computation. Some discrepancies are attributed to somewhat underestimated magnitude of ω_x in the present results, which is likely due to the limitation of the present isotropic eddy-viscosity turbulence model. Although the issue is involved, wave-induced effects displayed in the measurements are correctly reproduced in the present computation, which is described next.

Significant differences are observed between the Fn=0 and 0.316 results. On the forebody, differences are due to the wave crest and trough for both port and starboard sides, and are larger near the free surface and decay with depth. Effects of wave crest induce a broader region of reduced axial velocity and turn the direction of crossplane flow; and subsequently, effects of the wave trough induce increased axial velocity and downward crossplane flow. On the afterbody, the differences are due to the shoulder-wave system for both sides, and the stern-wave crest and, here again, are largest near the free surface and decay with depth. The crossplane flow displays significantly increased magnitudes. The boundary layer and wake thickness is reduced subsequently, especially near the free surface. For both forebody and afterbody, the keel and bilge vortices are larger for high Fn and somewhat altered in position.



Fig. 7 Comparison of computed pressure Cp contours (upright condition).

In most cases, the differences between the flows for high and low Fn can be directly related to the wave elevations and wave-induced pressure gradients. The differences for the U and W velocity components are shown to be correlated with ζ and $\partial \zeta / \partial X$, respectively, and those for V velocity component to be consistent with $\partial \zeta / \partial Y$. In some cases, the differences for U and W are shown to be consistent with the variations in $\partial Cp / \partial X$ and $\partial Cp / \partial Z$, respectively. The Cpdifferences are correlated with those for U. In conclusion, the wave-induced effects on mean-velocity and pressure fields are significant where the amplitudes of free-surface are large, and most of the wave influences can be explicated as a result of the wave elevations and wave-induced pressure gradients.



Fig. 8 Comparison of computed axial-vorticity ω_x contours (upright condition).



Fig. 9 Comparison of computed crossplane flow vectors V-W (upright condition).

4.3 Surface pressure and frictional streamlines

In this section, wave influences on surface pressure field and frictional streamlines are discussed. The model and computational conditions are same as those of the previous section.

The surface-pressure contours for Fn=0 and 0.316 are shown in Figs.10 and 11, respectively. Differences

162

are clear near the free-surface, i.e., the wave influences on surface-pressure field are significant near the waterline and decay with depth. Fig.12 compares the computed frictional streamlines between Fn=0 and 0.316. In contrast to significant differences of surfacepressure field near the waterline, the general aspects of separation pattern are similar between the two results, which is due to a fact that most of three-dimensional separations occur near the keel and bilge, where wave influences are relatively small.



Fig. 10 Surface-pressure Cp contours for Fn=0 (upright condition).



Fn=0.316 (upright condition).

On the forebody port side, a pocket of low pressure region is observed near the bilge, which causes the flow to converge towards the keel (denoted as A in the figure), and partially converges at the keel and separates to generate forebody keel vortex, and forebody bilge vortex in the location slightly downward (B). On the other hand, on the afterbody starboard side, low pressure region is located near the stern bilge, which causes the streamlines to converge towards the stern bilge, and meet the streamlines from port side and finally form the opened-type separation pattern, that results in generation of afterbody-bilge vortex (C). Streamlines on afterbody portside partially converge at the keel and separate to generate afterbody-keel vortex (D). The above-mentioned locations A through D are in good agreement with those of vortex generation displayed in mean-flow measurements 1),2).



of axial vorticity between for Fn=0 and 0.316 (upright condition).

Fig.13 shows comparison of the cross-sectional integrals of ω_x between Fn=0 and 0.316. The gradient of values indicates generation and decay of ω_x in total at the cross section. For both results, increases of the value are significant near the bow and stern, which is consistent with the above-discussed formation

-290-

of forebody and afterbody keel and bilge vortices; and the values decrease in the wake region due to fluid viscosity. It is noteworthy that the value is generally larger for Fn=0.316, which leads to a fact that wave-induced effects increase the cross-sectional total of axial vorticity. That agrees with implication from inviscid-flow method, e.g., results of the author ²⁰⁾, in which increase of doublet strength on wake-sheet due to wavemaking effects was indicated.



Fig. 14 Mean-flow solutions for Fn = 0.310 ($Rn = 5.2 \times 10^6$, sinkage-trim-heel condition).

4.4 Hydrodynamic forces

In this section, wave influences on hydrodynamic forces are discussed. Two computational conditions are selected, i.e., Fn=0 and 0.310, which are compared with measurements of Fn=0.1 and 0.310, respectively. The computational Rn are 1.7×10^6 and 5.2×10^6 , which are nearly same as those of experiment. The yaw angle $\beta = 10^\circ$ is again selected so that the discussions can be correlated with those of the earlier sections. In the experiment ^{1),2)}, sinkage σ , trim τ , and heel η were set free and the values were measured. In the present computation for Fn=0.310, model condition is fixed to be same (σ,τ,η) as those of the measurements, i.e., about $(0.15,-0.03,5.4^\circ)$. See reference ^{1),2)} for the definition and details. For computation for Fn=0, ship is fixed upright, since experimental (σ,τ,η) are found to be small such that upright double-model simulation can be applied.

Fig.14 shows the cross-sectional contours of meanvelocity and pressure fields for the present Fn=0.310case. It is seen that overall flow features discussed for upright case hold true for the present condition, e.g., keel and bilge vortex formation and wave influences on mean-velocity and pressure fields. Fig.15 provides comparison of longitudinal and side force distributions, i.e., dC_x/dX and dC_y/dX , respectively, and yaw moment distribution dCM_z/dX , in which the differences between Fn=0 and 0.310 can be related to wave-induced effects. The figure also includes wetted girth length distributions, whose integral indicates larger wetted surface area for Fn=0.310 with the given running attitude. Note that forces and moment are divided by $0.5\rho U_0^2 S_0$ and $0.5\rho U_0^2 S_0 L$, respectively, to be coefficients C_x , C_y , and CM_z , where S_0 is designed wetted surface area.



Fig. 15 Comparison of $d(C_x, C_y, CM_z)/dx$ and wetted girth-length (Girth) distributions between for Fn = 0 and 0.310.

The differences of dC_x/dX between Fn=0 and 0.310 are obvious especially on forebody and near the stern, that is clearly due to wave effects. Also, significant differences are shown for dC_y/dX and dCM_z/dX especially on the forebody, i.e., the wave influences are significant in the region most of the sideforce and yaw moment are generated. Fig.16 shows comparisons of integral variables. As compared to (C_x, C_y, CM_z) for low Fn, those values for high Fn are about (39%,25%,70%) larger in the present results, and about (32%,25%,70%) larger in the measurements. Overall agreements of the trends between the computations

-291-

and measurements are satisfactory although magnitudes of the values are, in general, slightly under predicted in the computations.

For both computations and measurements, quantitative differences due to wave-induced effects appear to be significant for all variables, and are found to be the largest for yaw moment, that is likely due to the change in center of effort. This leads to an important conclusion that wave effects must be considered for accurate prediction of hydrodynamics forces and yaw moment, especially for ships operated at high Fn in yaw condition.



Fig. 16 Comparison of hydrodynamic forces between the present computation and measurements.

4.5 Wave-induced vortices and prediction of wave breaking

Inclusion of wave breaking effects is of interest in the present problem. Although in the present theory, the wave breaking is not directly modeled in the freesurface boundary conditions, the present results indicate similar breaking-wave induced effects, i.e., waveinduced vortices, as those displayed in the measurements. In this section, discussion is made on the details.

Longo et al. ^{1),2)} observed in their experiments for high Fn (=0.316) that wave-induced vortices are evident on the port-side forebody which initiates near the bow underneath the breaking bow wave. Fig.17 shows local mean-flow filed at X=0.1 of the present results for same condition as the experiments. In the regions denoted as A and B, negative and positive values of ω_x are indicated, respectively. The crossplane flow V- W show vortical motion beneath the free-surface with high curvature, which is denoted as C in the figure. It is also seen that the axial-velocity (U) is decelerated in the region. The vortices are obviously generated due to the wave-induced pressure gradient, and the high curvature of free surface can be correlated with a stage just before wave breaks. The wave breaking subsequent to it would be simulated if that is correctly modeled in the free-surface boundary conditions.



Alessandrini, et al. ²⁵⁾ presented a method to predict wave breaking, by using a criterion proposed by Subramani et al. ³⁰⁾, i.e., wave breaks for $|\kappa\zeta| \ge 0.5$, where κ and ζ are free-surface curvature and elevation, respectively. Fig.18 shows regions of $|\kappa\zeta| \ge 0.5$ associated with wave field in the bow region of the present results. The high $|\kappa\zeta|$ regions are located near the FP (denoted as A and B in the figure), and the extent in portside coincides with the region of steep bow-wave crest indicated in the free-surface contours, all of which show some similarity with the measurements ^{1),2)}. The modeling of wave breaking has not been pursued in the present study, but is still of interest for future work.

-292-

Journal of The Society of Naval Architects of Japan, Vol. 186



ig. 18 Computed wave field near the bow $(Fn=0.316, Rn = 5.3 \times 10^6, \text{ upright condition}).$

5. Conclusion

This paper concerns an investigation of wave influences on viscous flow around a ship in steady yaw motion, which is based on detailed analysis of numerical solutions from a RaNS equation method developed by the present author. The results are presented for the Series 60 $C_B=0.6$ at yaw angle 10°, and validated through wherever possible comparison with experimental data for high and low Fn, in which satisfactory agreements are demonstrated for the local region wave field, the mean-velocity and pressure fields, forces, and yaw moment. Subsequently, the discussions are focused on identification of salient feature of wave-included effects on flows, including some specific aspects of flow for wave breaking and wave-induced vortices.

For the condition considered in the present study, wave-included effects on mean-velocity and pressure fields are significant for both sides of the hull where the amplitudes of free-surface are large, and most of the wave influences can be explicated as a result of the wave elevations, and wave-induced pressure gradients. The magnitudes of axial vorticity are clearly increased due to wave effects. Also, considerable differences due to wave effects are shown for force and moment distributions especially on the forebody, where most of the sideforce and yaw moment are generated. Particularly, wave influences on yaw moment are remarkable, which is about 70 % differences between with/without wave cases. Those leads to an important conclusion that wave effects must be considered for accurate prediction of hydrodynamics forces and yaw moment, especially for ships operated at high Fn in yaw condition.

Although the present method appears to be capa-

ble for investigation on the present main interests, improvements are expected for more detailed investigation on Reynolds-stress field with consideration of higher order turbulence model and comparison with data recently obtained by Suzuki et al. ³¹⁾. In addition, extension for dynamic simulation is of great interest, and currently in progress.

6. Acknowledgments

The author wishes to thank Profs. Himeno and Ikeda, and Dr. Katayama of Osaka Prefecture University for their valuable comments and encouragement.

References

- Longo, J.: Yaw effects on model-scale ship flows, Ph.D. Thesis, The Department of Mechanical Engineering, The University of Iowa, Iowa City, IA, 1996.
- Longo J. and Stern, F.: Yaw effects on modelscale ship flow, Proc. Twenty First Symposium on Naval Hydrodynamics, June 25, 1996, Trondheim, Norway, 1996.
- Ueno, K., Ueno, J., Hosoda, T., and Maeda, M.: Some Experiments of Yawing Effect on Ahead Resistance of Ships, Kyushu University Memoirs, Vol. 22, No.1, Fukuoka, Japan, pp. 37-45., 1962.
- Ueno, K., Egi, K., Kondo, K., and Yamaguchi, M.: Further Experiments of Yawing Effect on Ahead Resistance of Ships, Kyushu University Memoirs, Vol. 23, No.3, Fukuoka, Japan, pp. 169-187., 1964.
- Noshio, S., Tanaka, I., and Ueda, H.: Study on Separated Flow around Ships at Incidence, J. Kansai Society of Naval Architects, No. 210, pp. 9-17, 1988.
- 6) Hasegawa, Y., Okuno, T., Yamasaki, R., Nakanishi, and Y., Tanaka, N: Investigation into Flow Field around Fore Bottom of a High Speed Ship at Inicidence, J. Kansai Society of Naval Architects, No. 211, pp. 41-46, 1989.
- Okuno, T., Tanaka, N., and Hasegawa, Y.: Flow Field Measurement around Ship Hull at Inicidence, J. Kansai Society of Naval Architects, No. 212, pp. 67-74, 1989.
- 8) SR221 Final Report, Shipbuilding Research Association of Japan, 1996.

166

-293-

- Fuwa, T.: Hydrodynamic Forces Acting on a Ship in Oblique Towing, J. Society of Naval Architects of Japan, Vol. 134, pp. 135-147, 1973.
- Mizoguchi, S.: Calculation of Flow with Three-Dimensional Separation Vorticities around Ships, J. Kansai Society of Naval Architects, Japan, No. 188, pp. 57-66, 1983.
- Matsumura, K., Tanaka, I., Oki, T. and Kishi, S.; On the Nonlinear Lift Characteristics of Slender Bodies at Incidence, J. Society of Naval Architects of Japan, Vol. 154, pp. 283-293, 1983.
- 12) Tahara, Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, J. Kansai Society of Naval Architects, Japan, No. 226, pp. 63-76, 1996.
- Izumi, K, Sato, T, and Miyata, H.: CFD Simulation of Maneuvering Motion for Blunt Ships, J. Society of Naval Architects of Japan, Vol. 184, pp. 55-61, 1998.
- 14) Nonaka, K.: Free Surface Effects on the Side Force and Moment Acting on a Ship Hull with a Drift Angle, J. Society of Naval Architects of Japan, Vol. 138, pp. 178-187, 1975.
- 15) Hori, T., Matsumura, K., and Tanaka, I.: On the Lateral Force Caused by Wave Generation Acting on the Ship Hull with Steady Drift Angle, J. Society of Naval Architects of Japan, Vol. 159, pp. 9-22, 1986.
- 16) Song, W.S.: Wave-making Hydrodynamic Forces Acting on a Ship with Drift Angle and Wave Pattern in Her Neighborhood, J. Kansai Society of Naval Architects, Japan, No. 211, pp. 25-32, 1988.
- 17) Maruo, H., and Song, W.S.: Numerical Appraisal of the New Slender Ship Formulation in Steady Motion, Proc. Eighteenth Symposium on Naval Hydrodynamics, Ann Arbor, pp. 239-257, 1990.
- 18) Nakatake, K., Komura, A., Ando, J., and Kataoka, K.: On the flow Field and the Hydrodynamic Forces of an Obliquing Ship, J. Seibu Society of Naval Architects, No. 80, pp. 1-12, 1990.
- 19) Xia, F. and Larsson, L.: A Calculation Method for the Lifting Potential Flow Around Yawed, Surface-Piercing 3-D Bodies, Proc. Sixteenth Symposium on Naval Hydrodynamics, Berkeley, pp. 583-597, 1986.

- 20) Tahara, Y.: A Boundary-Element Method for Calculating Free-Surface Flows Around a Yawed Ship, J. Kansai Society of Naval Architects, Japan, No. 218, pp. 55-67, 1992.
- 21) Rosen, B.S., Laiosa, J.P., Davis, W.H. and Stavetski, D.: SPLASH Free-Surface Flow Code Methodology for Hydrodynamic Design and Analysis of IACC Yachts, Proc. Eleventh Chesapeake Sailing Yacht Symposium, Annapolis, MD, pp. 35-49, 1993.
- 22) Iwasaki, M, Suzuki, K., and Kai, H.: Numerical Analysis of Free Surface Flow around Sailing Yacht by Means of Panel Method -1st Report: Calculation of Hydrodynamic Forces in Upright Condition without and with Leeway-, J. Kansai Society of Naval Architects, Japan, No. 230, pp. 65-74, 1998.
- 23) Campana, E.F., Di Mascio, A. and Penna, R.: CFD Analysis of the Flow Past a Ship in Steady Drift Motion, Proc. Third Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, pp. 151-159, 1998.
- 24) Tahara, Y., Longo, J., Stern, F. and Himeno, Y.: Comparison of CFD and EFD for Series 60 C_B=0.6 in Steady Yaw Motion, Proc. Twenty Second Symposium on Naval Hydrodynamics, July 1998, Washington D.C., USA, 1998.
- 25) Alessandrini, B.A., and Delhommeau, G.: Viscous Free Surface Flow Past a Ship in Drift and in Rotating Motion, Proc. Twenty Second Symposium on Naval Hydrodynamics, July 1998, Washington D.C., USA, 1998.
- 26) Tahara, Y.: Evaluation of a RaNS Equation Method for Calculating Ship Boundary Layers and Wakes Including Wave Effects, J. Society of Naval Architects of Japan, Vol. 180, pp. 59-80, 1996.
- 27) Stern, F., Paterson, E.G. and Tahara, Y.: CFDSHIP-IOWA: Computational Fluid Dynamics Method for Surface-Ship Boundary Layers, Wakes, and Wave Fields, IIHR Report, No. 381, Iowa Institute of Hydraulic Research, Iowa City, IA 52242, USA, 1995.
- 28) Coleman, H.W. and Stern F.: Uncertainties and CFD Code Validation, ASME J. Fluids Engineering, Vol. 119, pp. 795-803, 1997.

-294-

- 29) Zhang, Z.J. and Stern, F.: Free-Surface Wave-Induced Separation, ASME J. Fluids Engineering, Vol. 118, pp. 546-554, 1996.
- 30) Subramani, A.K., Beck, R.F., and Schultz, W.W.: Suppression of wave-breaking in nonlinear water wave computations, Thirteenth International Workshop on Water Waves and Floating Bodies, Delft, March 1998.
- 31) Suzuki, H., Miyazaki, S., Suzuki, T., and Matsumura, K.: Experimental Study on Turbulent Flow Field around an Oblique Ship Model, J. Kansai Society of Naval Architects, Japan, No. 230, pp. 123-132, 1998.

アメリカ杯レース艇のバラストバルブ開発 - 特に基本低抵抗形状の検討について - *1

正会員田原裕介*²,正会員吉田豊*³,正会員西田隆司*⁴ 正会員松井亨介*⁵,上野一郎*⁶

Development of Ballast Bulb for IACC Sailing Yacht - Especially for Investigation on Basic Low Drag Form -

By Yusuke TAHARA (Member), Yutaka YOSHIDA (Member) Ryuji NISHIDA (Member), Ryosuke MATSUI (Member) and Ichiro UENO

This paper provides an introduction of ballast bulb development for International America's Cup Class (IACC) Sailing Yacht, that had been associated with the hull/appendage design project supported by Nippon Challenge Syndicate for the 30th America's Cup. In this report, main emphasis is especially placed on investigation on basic low drag form, which was one of research activities carried out by appendage design team. An overview is given for the Computational Fluid Dynamics design tool in association with the experimental devices and procedures, then discussions are focused on important design aspects for prediction of laminar-to-turbulent flow transition occurs on the body at the full-scale-level Reynolds number.

Keywords : America's Cup Sailing Yacht, Ballast Bulb, RaNS Equation Method, Tank Test, Laminar-to-Turbulent Flow Transition

1. 緒言

本論文は,第30回アメリカ杯ヨットレースにおい て,ニッポンチャレンジ・アペンデージ開発チーム(以 下NC-ADTと略称)で行われた,バラストバルブの開 発について述べるものである.アメリカ杯レース艇 (International America's Cup Class Yacht:以下IACC ヨット)は,一般のヨットより遥かに大きい面積のセー ルを有し,そのヒーリングモーメントに対して安定性 を確保するために,大重量のバラストバルブを必要と する.バルブの設計要件は低重心,モーメントバラン スなど数多くあるが,最も基本的な要素は極力小さな 抵抗特性を有することである.主船体の場合と異なり, 実スケールにおける層流-乱流遷移点の推定が困難であ るなどの問題があり,小型模型試験によって実船搭載 時の性能を正確に予測することが難しい(関連文献に ついては著者ら¹⁾や文献²⁾⁻⁵⁾を参照).

IACCヨット搭載用バラストバルブの最適形状に関 しては、現在のIACCルールが適用された1992年アメ リカ杯以降、参加各シンジゲートによって、乱流型(特 化された例として1995年第29回アメリカ杯における Team Dennis Connerなど)か層流型、もしくはその 中間的な形状かが検討されてきた.しかしながら、前 回アメリカ杯(1995年第29回大会)の時点においても、 最適形状に関する明確な設計思想を持つシンジゲート は少なかったと思われる.

ー方NC-ADTでは、レイノルズ平均ナビエ・ストークス方程式法(Reynolds-averaged Navier Stokes Equation Method: RaNS法)に基づくストラット・バ

^{*1} 平成12年5月25日関西造船協会春季講演会において講 演,原稿受付 平成12年6月9日

^{*2} 大阪府立大学大学院工学研究科

^{*3} 三菱重工業(株)(研究当時 大阪府立大学大学院工学研 究科)

^{*4} 石川島播磨重工業(株)(研究当時 大阪府立大学大学院 工学研究科)

^{*5 (}株)三井造船昭島研究所

^{*6 (}有)シールズ

ラストバルブ設計ツールの開発を行うとともに(初期 の報告として文献¹⁾),回流・曳航水槽実験におけるバ ルブ流体力の評価法を検討してきた.航走状態で想定 されるバルブのレイノルズ数(Rn)は10⁷のオーダーで あり,本研究における検討の結果,現在検討対象とな る一般的なバルブ形状に関しては,その抵抗成分の殆 どを摩擦抵抗が占めることや,摩擦抵抗の評価におい ては層流域の影響が無視できなことに加え,より低抵 抗な形状を検討する場合は,層流-乱流遷移の考慮が重 要になることを明らかにした.

本稿の主眼は、NC-ADTの設計活動の一環として行われた、直進状態(ゼロ迎角)における基本低抵抗形状の検討に関する報告である.実際のIACCヨットは数度の斜航角をもって航走するため、NC-ADTでは横力発生時の抗力特性の検討も行っており、その詳細は将来報告する予定である.本研究においては、RaNS法を実スケールレベルRnで適用し、実用的な層流-乱流遷移判定法を導入することにより、実験で得られた供試模型の抵抗値を正確に予測することが可能となった.またそのRaNS法を用いて設計した形状を実験で検証した結果、RaNS法で予測されたとおりの低抵抗特性を有する形状であることが確認できた.

2. 計算手法および実スケールレベル実験手法の概要

2.1 計算手法

ここでは、本研究で用いた RaNS 法の概要を述べる. 基礎方程式は3次元非定常,非圧縮流体に関する RaNS 方程式である.数値計算においては,基礎方程式を数値 的に生成した非直交曲線座標系に変換し、さらに輸送 方程式を有限解析法により離散化する.また速度場/圧 力場結合にはPISOタイプワンステップ法を採用する. RaNS法は著者らの研究¹⁾で開発したものと基本的に 同一であるが、今回実スケール Rnへの適用、および 層流-乱流遷移影響を実用上十分な精度で考慮できるよ うに、遷移判定型乱流モデルの導入や計算格子の改良 を行った.計算格子の一例をFig.1に示す.オーダー 10^7 のRnにおける計算にも対応できるように,壁面近 傍の最小格子間隔はy⁺≈1になるように設定した..な お計算は任意のバルブ形状だけでなく、フラップ作動 時を含む任意のストラット形状にも対応できる.数値 計算法の詳細は著者らの文献^{1),6)-8)}を参照されたい.

本研究では、後述する層流-乱流遷移を考慮した計算 に先立ち、モデルスケール Rnにおいて強制的な層流 -乱流遷移を仮定した場合の RaNS 法の精度を確認した. 同様な検討は以前も行っているが¹⁾、今回は供試模型 数を増やし、より多様な形状への適用性を調査した. 実験は回流水槽(神戸商船大学および大阪府立大学)で 行い、供試模型には前回のアメリカ杯レースに使用さ



Fig. 1 Overview of computational grid.



Fig. 2 Comparison of bulb/strut resistance between 1/5-scale computations and experiments (condition: forced laminar-toturbulent flow transition, $Rn \approx 2 \times 10^6$).

れた形状を含め、形状や全長の異なる1/5スケール・ モデルを用いている.なお実験においては模型先端近 傍に乱流促進を施し、計算においては同位置から乱流 モデルを用いることによって実験・計算条件を整合さ せた.また実験では同一形状のストラットを用い、さ らに水面板を設置することによって自由表面影響を除 去している.一定流速(*Rn* ≈2×10⁶)・直進状態にお ける実験値と計算値の比較をFig.2に示す.計算値はバ ルブ部のみの評価値であるため、ストラット部および 水面板の固有抵抗を含む実験値との間には定量的な開 きがあるが、実験で得られたバルブ間の抵抗値順位は、 計算で予測された抵抗値順位と全て一致している.こ れは、ここで考慮した条件における、本手法の妥当性 を示すものと考えられる.

2.2 実スケールレベル実験手法

ここでは、実スケールレベルの実験手法の概要を述べる.実験は曳航水槽(三井昭島研究所)において行った. 供試模型は実船搭載を想定した1/2スケール・モデル であり、実船想定以上の速度で曳航することによって、

-297-



Fig. 3 Experimental facility (Akishima Lab.).

実スケールと同レベルの Rnを達成させた.実験状態 をFig.3に示す.同一形状のストラットを用い,また計 測値に含まれるストラット部の影響を同等とするため, バルブ上面から静止水面までの距離を一定とした.な お実船搭載時の状態と同様,ストラットおよびバルブ 表面の租度は実際と同レベルとなるように配慮した.

以下,数種の供試模型の中から特徴的な3つの形 状,すなわちType-A~Cに関する結果について議論す る.それらの供試模型の側面プロファイルをFig.4に 示す.Type-A~Cの体積は一定であり,断面形状の違 いによって全長や最大高さ・幅,さらに重心位置など が異なっている.実際と同様,全ての模型は重心低下 を目的としてやや扁平な断面を有する.Type-Aは今 回のバルブ開発で標準的としたもの,そしてType-C はいわゆる層流型である.全長はType-AとType-Bが ほぼ同一であり,Type-Cは他の2つに比べて細長い. Type-A,Cは上下対称であり,Type-Cの断面形状はよ り真円に近い.Type-Bは本研究で低重心特化型と定 義したもので,断面を上下非対称とすることによって より低重心化を図ったものである.

ー般に長さおよび体積が一定であれば、断面が真 円に近いほど表面積が小さくなり、抵抗特性としては 有利であるが、重心位置を下げる効果は小さくなる。 IACCヨットの場合は全排水量の8割近くがバラストバ ルブの重量であり、数センチでもバルブ重心を下げる ことによって、ヨット全体の重心低下に大きな効果が あるため、真円断面形状が用いられることは殆どない。

まず初期段階の検討として、本研究ではストラット 部の造波影響を調査した.ストラットのみを直進状態で 曳航し、計測した抵抗係数をFig.5に示す.なおCtは 全抵抗、CwはCtより相当平板の摩擦抵抗成分を差引 いたものである.またFnはストラットのコード長に基 づくフルード数、そしてCd*=Ct/Cd_M(or Cw/Cd_M) は得られた抵抗係数CtもしくはCwを一定の指標抵抗



Fig. 4 Side view of computational and experimental test models.



係数*Cd_M*で除した値である.*Fn*が増加するに従い, *Cw*はいわゆるスプレー抵抗係数に漸近する傾向が確認 できる.今回の実験で主な評価点としたのは*Fn*=2.5 近傍であり,その領域でのストラットの造波抵抗は比 較的安定していると判断した.

つぎに、バルブ没水深度を変化させることにより、 バルブ抵抗に及ぼす自由表面影響を調査した. Type-A に関し、バルブ没水深度(Strut Span)を変化させた場 合の、ストラット部のみ(Strut only(1))と、ストラッ トにバルブを装着した状態(Strut+Bulb(2))の抵抗値 の計測結果をFig.6に示す.計測は直進状態で行い、曳 航速度を4.5m/sに固定した. この状態でのストラット のコード長に基づくFnは約2.5である. また図中の抵 抗値(D_*)は、計測値Dを一定の指標抵抗値 D_M で除し た値である. Fig.6における(2)-(1)の傾向より、バル ブ没水深度を変化させた場合の、バルブ抵抗に及ぼす 自由表面影響を検討することができる.



Fig. 6 Influences of strut span length (depth of bulb top) on measured bulb/strut resistance.



mergence ratio (Hoerner⁹⁾).

Fig.6に示す結果より、バルブ没水深度が増すほど (2)-(1) は減少する傾向が確認できる.没水体の造波影 響については, Fig.7に示す関係⁹⁾が知られており, 没 水深度hと没水体全長Lとの比h/Lが1に近づくにつ れ、没水体流体力に及ぼす自由表面影響は急速に減少 する. Fig.6における最大没水深度のh/Lは約0.5であ り、この状態ではFig.7の値から推定しても、没水体流 体力に及ぼす自由表面影響は若干ながら存在すること がわかる.それはFig.6に示す実験値においては、最大 没水時に全抵抗の5%弱となっている.一方,今回の実 験における供試模型の体積は一定であること、また全 長差も顕著でないことより、最大没水深度において模 型間の抵抗値順位を評価する目的に関しては、自由表 面影響は無視できる範囲にあると判断した.以下に示 す実験値は、全てFig.6に示す最大没水深度における ものである、さらに今回は、ストラット部の抵抗は各 供試模型に関する実験において同等であると仮定し,



Fig. 8 Comparison of measured bulb resistance for Type-A through C bulbs.



with flat-plate values.

ストラット部のみの曳航試験で得られた抵抗を,全抵 抗から差引いたものをバルブ抵抗と定義した.

3. 層流-乱流遷移判定法の検討

本研究では、Type-A~Cに関する実験値を用いて、 形状差による層流-乱流遷移の違いを調べ、層流-乱流遷 移判定法の検討を行った.ここではその概要を述べる. 直進状態で曳航速度を変化させた場合の、バルブ抵抗 値の変化をFig.8に示す.図中の最高流速(4.5m/s)に おけるバルブ Rnは約1×10⁷になる.Fig.8に示す流 速全域において、低重心特化型のType-Bの抵抗値が 最も高い.一方、Type-Aと層流型のType-Cは流速に よって抵抗順位が入れ替わっている.Type-Cは低速域 ではType-Aより低い抵抗値を示すが、高速域では明 らかにより高い抵抗値を示す.前節で検討した結果に 基づき、バルブ抵抗の抵抗順位における自由表面影響 は無視できると仮定すれば、これはバルブ粘性抵抗の 特性の変化によるものと考えられる.

Fig.9はType-A に関するFig.8の結果を*Rn*基軸で示したものである.同図にはSchoenherr式,層流での



Fig. 10 Functions f_1 and f_2 for detection of laminar-to-turbulent flow transition (Granville¹⁰).

平板摩擦抵抗である Blasius式,層流から乱流への遷移を考慮した平板摩擦抵抗である Prandtl-Schlichting 式の値も示す.Prandtl-Schlichting式では遷移レイノ ルズ数 R_{Xt} を仮定する必要があり,図中の値は $R_{Xt}=1$ ×10⁶および $R_{Xt}=3$ ×10⁶の結果である.計測された Type-Aの抵抗値は Schoenherr式の値を下回り,また $R_{Xt}=3$ ×10⁶の Prandtl-Schlichting式の値に似た傾向 を示している.計測値には圧力抵抗成分も含まれており,実際の摩擦抵抗成分は図中の曲線をほぼ平行下方移動したものと考えられる.以上の結果は,本研究で 標準型としたType-Aにおいても,正確な摩擦抵抗の 推定には層流-乱流遷移を考慮する必要があることを示 している.

つづいて本研究では、実用上十分な精度を有する層 流-乱流遷移点判定法の検討を行った.まず検討を簡便 に行うために、Type-Aのサイドラインを用いた回転 体を考え、積分型の境界層方程式を解き、得られた境 界層パラメータを用いてGranville¹⁰⁾の方法により層 流-乱流遷移点の評価を行った.遷移点以前は層流境界 層として、また以後は乱流境界層として計算する.境



Fig. 11 Comparison of local shear stress distributions.



Fig. 12 Influences of control parameter C_{MUTM} on computed bulb resistance.

界層計算の方法は,遷移点以前は層流計算とした点以 外,著者らが以前行った自由表面/境界層相互干渉問 題に関する研究¹¹⁾のものとほぼ同一である.

一方, Granvilleの遷移点判定法は,軸対称物体を 用いた風洞実験の結果に基づいている.その方法で は, $Rn=10^{6}\sim 10^{7}$ の領域において,不安定点での R_{θ} (運 動量厚スケール・レイノルズ数)と遷移点での R_{θ} の 差 $(R_{\theta})_{t} - (R_{\theta})_{n}$ を,幾何パラメータ κ の関数 $f_{1}(\kappa)$ で 与える.さらに境界層外端流速の勾配と運動量厚 θ に基づくパラメータ λ を定義し,不安定点 $(R_{\theta})_{n}$ は関 数 $f_{2}(\lambda)$ を用いて検出する.ここで関数 $f_{1}(\kappa)$ および $f_{2}(\lambda)$ はFig.10に示すような実験式である.本研究で は,Granvilleの遷移点判定法を上述した境界層計算法 に導入し,Type-Aのサイドラインの回転体について 境界層計算を行った.直進状態において $Rn=1 \times 10^{7}$



Fig. 13 Comparison of computed local shear stress contours (for Type-A bulb).

で求めた局所摩擦係数 C_{τ} をFig.11にB.L.T.として示 す. 図中X=0,1がそれぞれバルブ先端および後端に対 応する. $X=0.4\sim0.5$ の領域においてB.L.T.の C_{τ} が急 変する部分があり、これがGranvilleの遷移点判定法に より検出された層流-乱流遷移点と考えられる.

つぎに本研究では、RaNS法に簡便な方法で導入で きる,層流-乱流遷移点判定法を検討した.本研究で 開発した RaNS 法には Baldwin-Lomax (BL) 乱流モデ ル¹²⁾が搭載されており、今回このBLモデルの修正を 試みた. Baldwin¹²⁾らは, 層流-乱流遷移点を近似的に 判定する方法として,壁面法線方向の渦動粘性係数 ν_tの分布を調べ、その最大値によって乱流遷移を判定 する方法を提案している.具体的には, ν_tの法線方向 分布の最大値が判定値_レ·C_{MUTM}以下の場合を層流で あるとし, Baldwinらは平板境界層の結果に基づいて C_{MUTM}=14を提案している.本研究では,この定数 C_{MUTM}の再検討を試みた. Fig.11には上述した回転 体に関し、*C_{MUTM}*=14~30によってRaNS法で計算し たC_r分布も示す.計算条件は上述したB.L.T.の場合 と同一である. RaNS法の結果においてもCrが急変す る領域があり、これが判定された層流-乱流遷移である と考えられる.前述した境界層計算の場合と異なり, 遷移は段階的に起こる様子が認められ、その領域は C_{MUTM}=25~30の場合において, B.L.T.の遷移点の 近傍となる.

 C_{MUTM} の変化が RaNS法の解に与える影響を調べるために、Type-Aに関して C_{MUTM} =14~30の範囲で計算を行った. 直進状態・Rn=1×10⁷において、得られた抵抗係数(Cd*)をFig.12に示す. C_{MUTM} が増加



Fig. 14 Comparison of computed local shear stress contours (for Type-C bulb).

するに従い層流領域が拡大し、摩擦抵抗が減少する.一 方、圧力抵抗における C_{MUTM} の影響は比較的小さい ことがわかる.Type-A,Cに関し、直進状態・ $Rn=1 \times$ 10^7 において、RaNS法で求めた物体表面上の C_r 分布を Fig.13およびFig.14に示す.図中、物体前半部の暗領 域は C_r 値が低い部分を示し、層流と考えられる領域を 大まかに表している.両模型に関し、乱流に遷移した後 の再層流化に相当すると思われる状態も確認できる.

Fig.13およびFig.14の結果では、Type-AとCでは、 Type-Cの乱流領域の方が広いことが認められ、また 算出した抵抗値もType-Cの方が高いことを確認した. 一方、図中には示していないが、より低いRn(Fig.8に おける曳航速度3m/s相当)における計算を行った結果 では、Type-Cの乱流領域の方がType-Aのものより狭 く、また抵抗値も低くなっており、これはFig.8に関し て述べたType-AとCの抵抗値の相対関係と合致した ものである.層流型形状は、適用流速によっては遷移 点の前方移動に伴って乱流域が急速に拡大し、抵抗性 能の劣化が顕著になる特徴があると考えられる.

最後に、Type-A~Cに関し、 C_{MUTM} =30を用い、 RaNS法によって予測した抵抗値の精度を検討した結 果について述べる.これまでの議論と同様に、直進状 態における計算値と実験値の比較をFig.15に示す.実 験値は曳航速度を固定し(平均的なRnは約1×10⁷)、 計算条件は実験状態に適合させた.本RaNS法は、実 験値の抵抗値順位を正確に予測していることが分かる. また計算結果を検討した結果、物体後端近傍における バブル型剥離などは見られず、抵抗成分の殆どは摩擦 抵抗であることも確認した.すなわち、本研究で主対



Fig. 15 Comparison of bulb resistance between full-scale-level computations and experiments (condition: $Rn \approx 1 \times 10^7$, natural laminar-to-turbulent flow transition).



Fig. 16 Comparison of bulb resistance between full-scale computations and experiments (condition: $Rn \approx 1.5 \times 10^7$, natural laminar-to-turbulent flow transition).

象とするようなバルブの低抵抗形状を検討する場合に は,層流-乱流遷移を考慮した,高精度な摩擦抵抗の推 定法が必要であることが明らかとなった.一方,全乱 流状態を仮定した計算(*C_{MUTM}=0*)も行い,その結果 を検討した結果,抵抗値は全体的に実験値より高くな り,加えて実験値に見られる模型間の抵抗値関係を正 確に表現できないことが分かった.

4. 実設計への応用

ここでは、前節で述べた手法を実設計に応用した結果 について議論する.デザイン・ポイントRnは約1.5× 10^7 であり、実船スケールRnとほぼ一致する. RaNS 法を用いて抵抗値を予測しながら、低抵抗特性を重視 して重心位置を変えた3種の形状を設計し、最終的に 曳航水槽において、その形状を評価した. 直進状態に



Fig. 17 Comparison of bulb length of the 30th America's Cup entry boats.

おける結果をFig.16に示す. 図中Type-N1はType-A とほぼ同傾向の形状を維持しつつ,層流領域の拡大を 図ったものであり,Type-N2,N3は重心位置をやや上げ, 断面形状を若干真円に戻す方向でより低抵抗型を目指 したものである.実験値と計算値は定量的・定性的に も良く一致しており,本研究で開発した手法の有効性 を示すものと考えられる.

一方, 第30回アメリカ杯ヨットレースにおいて, ニッ ポンチャレンジ所属のレーサーヨットに搭載されたバ ラストバルブは、本研究で開発した手法を用い、また 冒頭に述べたように有迎角時の抵抗特性も勘案して設 計され、実際にセール番号 JPN52のボートなどに搭載 された. 上位チームのアペンデージの殆どは公開され ており、それらの形状との比較によって設計思想の妥 当性が確認できる.一例として、上位チームのバルブ 長の比較をFig.17に示す.レース参加艇全体では、バ ルブ長の分散範囲は比較的広く、最適長さに関する設 計思想は明確でないチームもあったと思われる.今回 設計したバルブ長は、上位チームで採用されたものと ほぼ同レベルの範囲にあった.また形状を比較した結 果,前回アメリカ杯(1995年第29回大会)のサンディエ ゴ沖から、今回のオークランド沖の海況への移行に際 し、重心位置の思想や、低抵抗型バルブを狙った設計 思想に関しては、同じ方向性にあったことが分かった.

最後に,本論文においては詳細を述べるに至らな かった,アペンデージの斜航時特性に関し,バルブ後 端近傍に装着されるウイングに関して述べる.Fig.18 は大阪府立大学回流水槽で行った,バルブ/ウイング 接合部の近傍流場の,油膜法による可視化結果である. 流場の大まかな調査が目的であったため,設定Rnは 約700,000とし,物体先端部近傍には乱流促進を施し た.図中,バルブ/ウイング接合部近傍の流場は,迎 角によって大きく異なっていることが分かる.実際の レースでは、ウイング設定角の微細なずれが、アペン デージ性能に大きく影響することが知られている.

ウイング効果に関しては、旧12 Meterルール時代に ウイングレット・キールが登場して以来、比較的多く の議論が行われた.しかしながら、現IACCヨットの ウイングは、通常バルブの後半部に装着され、この場 合のウイングの効能については、完全には解明されて いない部分がある.ウイングは主に風上帆走時に有効 であるが、反して風下帆走時には抵抗増加の要因とな りえるため、レース全体を考えた有効性の検討が必要 である.NC-ADTにおいてはレース艇の運動に伴う効 果も含め、ウイング効果に関する詳細な検討も進めて おり、その成果は将来報告する予定である.

5. 結言

本論文は,第30回アメリカ杯ヨットレースにおい て,ニッポンチャレンジ・アペンデージ開発チームで 行われた,バラストバルブの開発について述べるもの である.その設計活動の一環として行われた研究の中 から,本稿では,特に直進状態における基本低抵抗形 状の検討に関して報告した.横力発生時の抗力特性の 検討については,将来報告する予定である.本研究で 得た結果を以下に要約する.

(1) 現在検討対象となるバルブ形状に関しては、その抵抗成分の殆どを摩擦抵抗が占めること、また摩擦 抵抗の評価においては層流域の影響が無視できないこ と、さらに低抵抗形状を検討する場合は、層流-乱流遷 移の考慮が重要になることを明らかにした。

(2)本研究において開発したRaNS法を実スケール レベル Rnで適用し,実用的な層流-乱流遷移判定法を 導入した結果,実験で得られた供試模型の抵抗値を正 確に予測することが可能となった.またそのRaNS法 を用いて設計した形状を実験で検証した結果,RaNS 法で予測されたとおりの低抵抗特性を有する形状であ ることが確認できた.

(3)本研究の手法によって開発したバルブ形状を,第 30回アメリカ杯ヨットレースにおける上位チームのも のと比較した結果,多くの観点で同方向の設計思想に 基づくものであったことが分かった.

最後に、本研究を行うにあたり、種々のご助言ご協 力をいただきました大阪府立大学大学院工学研究科・ 姫野洋司教授、東京大学大学院工学研究科・宮田秀 明教授、ならびに大阪大学大学院工学研究科・戸田保 幸助教授に感謝の意を表わします.なお本研究の一部 は(株)ニッポンチャレンジ開発技術委員会との共同研 究として行われました.(有)ACT・金井亮浩博士(研究 当時、東京大学大学院工学系研究科助手)をはじめ技術 委員会の委員各位、さらに東京大学、神戸商船大学、





Fig. 18 Flow visualization near the wing/bulb junction.

ならびに大阪府立大学の学生各位のご協力とご助言に 謝意を表します.

参考文献

- 1) 田原裕介,吉田豊,西田隆司: RaNS法によるアメ リカ杯レース艇のストラット/バルブ近傍流場の 数値計算および実験との比較,関西造船協会誌,第 230号,1998,pp.141-146.
- Larsson, L. and Eliasson, R.E.: Principles of Yacht Design, A and C Black Publishers Ltd., London, 1994.
- 広嶋文哉,宮田秀明,秋元博路:CFD 航走シミュレーションによる帆走艇の設計法,日本造船学会論文集,第181号,1997,pp.15-23.
- 4) 片岡秀太郎,宮田秀明,金井亮浩:CFD 航走シミュレーションによる航走艇の設計法(第2報),日本造船学会論文集,第182号,1997,pp.113-120.

- 5) 今井新,内山幸樹:IACC帆走艇のキールシステム に関する研究,卒業論文,東京大学工学部船舶海洋 工学科,1994.
- Tahara, Y.: Computation of Viscous Flow Around Series 60 Model and Comparison with Experiments, J. Kansai Society of Naval Architects, Japan, No. 220, 1993, pp.29-47.
- Tahara, Y.: Computation of Boundary-Layer and Wake Flows around IACC Sailing Yacht, J. Kansai Society of Naval Architects, Japan, No. 224,1995,pp.1-11.
- Tahara, Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, J. Kansai Society of Naval Architects, Japan, No. 226, 1996, pp.63-76.
- 9) Hoerner, S.F.: Fluid-Dynamic Drag, Hoerner Fluid Dynamics, 1965.
- Granville, P.S.: The Calculation of the Viscous Drag of Bodies of Revolution, David Taylor Model Basin Report No. 849, Jul. 1953.
- 池畑光尚,田原裕介:船体まわりの自由表面流に及 ぼす境界層と伴流の影響,日本造船学会誌,第161 号, 1987, pp.51-59.
- Baldwin, B.S. and Lomax, H.: Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, 1978, pp. 1-8.

討 論

[討論] (金沢工業大学)増山 豊

セーリングヨットは、バラストキールを必要とする 風上帆走では横流れを伴います。今回の発表では直進 の場合のみについて述べられていますが、横流れを伴 う場合の優劣は異なってくるのではないでしょうか。 また横流れが境界層計算に与える影響が分かりました らご教示ください。

[回答]

本文中にも述べていますように、実際の設計におい ては、Leeway状態(通常5°以下のレベル)を考慮する とともに、他の設計要件(モーメントバランスなど)も 勘案していますので、最終的にはより多様な要件を満 足する形状が選択されました.一方、初期設計段階に おける直進時・低抵抗形状の検討は、重心位置やデザ イン・ポイントの変化に対する形状変化の傾向を把握 する目的に関し、極めて有益な基礎情報を提供します。

なお今回の研究における基本的なCFD手法はRaNS 法であり、Leeway状態における設計にもそれを使用し ています.そのため,ご質問の横流れが境界層計算に 与える影響については,ご提供できる直接的な情報が ないことをご了承ください.

Journal of Marine Science and Technology © SNAJ 2002

Comparison of CFD and EFD for the Series 60 $C_{\rm B} = 0.6$ in steady drift motion

YUSUKE TAHARA¹, JOSEPH LONGO², and FREDERICK STERN²

¹Department of Marine System Engineering, Osaka Prefecture University, 1-1 Gakuen-cho, Sakai 599-8531, Japan ²The University of Iowa, IIHR-Hydroscience & Engineering, College of Engineering, Iowa City, IA 52242-1585, USA

Abstract: This paper presents comparisons of computational and experimental fluid dynamics results for boundary layers, wakes, and wave fields for the Series 60 $C_{\rm B}$ = 0.6 ship model in steady drift motion. The numerical method solves the unsteady Reynolds-averaged Navier-Stokes and continuity equations with the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. The experimental and computational conditions, i.e., Froude numbers of 0.16 and 0.316 for the experiments, and Froude numbers of 0 and 0.316 for the computations, allow comparisons of low and high Froude number results, respectively, which allows an evaluation of Froude number effects and validation of the computational fluid dynamics at both low and high Froude numbers. This article gives an overview of this numerical approach, and the computational conditions and uncertainty analysis are described. Results are presented for the wave and flow fields, with emphasis on the important flow features of drift- and wave-induced effects in comparison with the experiments. Finally, conclusions from the present study are given, together with recommendations for future work.

List of symbols

AP, FP	after perpendicular, forward perpendicular
\hat{p}	piezometric pressure
ϕ	sampled variables or integrals
β	drift angle
ω_x	axial vorticity
$C_{\rm B}$	block coefficient
C_{p}	pressure coefficient (= $2\hat{p}/\rho U_o^2$)
Fr	Froude number (= U_o/\sqrt{gL})
g	gravitational constant
Н	total head $(=\sqrt{C_p + U^2 + V^2 + W^2})$
L	characteristic (ship) length

Address correspondence to: Y. Tahara

(tahara@marine.osakafu-u.ac.jp)

n	normal direction; time level
n	normal unit vector
р	static pressure; order of accuracy
r	radial coordinate; grid-refinement ratio
Re	Reynolds number (= $U_0 L/v$)
$S_{\rm b}, S_{\rm e},$ etc.	boundaries of the solution domain
U, V, W	mean velocity components
$U_{\rm E}, U_{\rm D},$ etc.	uncertainties
$U_{ m o}$	characteristic (freestream) velocity
X, Y, Z	Cartesian coordinates
$Y^{\scriptscriptstyle +}$	dimensionless distances (= $\operatorname{Re}U\tau Y_n$)
Y _n	distance normal to the wall surface
v	kinematic viscosity
Vt	eddy viscosity
ρ	density
ξ, η, ζ	body-fitted coordinates
ζ	free-surface elevation, residuals

Introduction

Although the drift condition is an approximation to the maneuvering ship condition,¹ it involves many features of interest that must be evaluated accurately for hull form design, e.g., asymmetric wave and flow patterns, breaking waves and breaking-wave wakes, wave- and body-induced vortices, spray, and bubble entrainment.^{2,3} In contrast to the straight-ahead (zero-drift) condition, the drift condition is dominated by strong crossflow effects, including forebody and afterbody keel and bilge vortices. The distinct vorticity in the flow field originates from strong drift-induced crossflow and large crossplane pressure gradients. For the high-Froude number (Fr) case, the wave-induced effects on the boundary layer and wake are significant owing to the larger wave amplitudes, especially on the windward side, than for the zero-drift case.

Recent advances in computational fluid dynamics (CFD) allow the application of Reynolds-averaged

Received: August 31, 2001 / Accepted: March 25, 2002

Navier–Stokes (RaNS) equation methods to the present problem, including maneuvering motion.⁴⁻¹² However, in most cases, the wave effects are not considered in the theory, where the free surface is treated as a symmetry plane. On the other hand, considerable effort has been focused on development of the RaNS equation method for calculating viscous free-surface flows, and the recent status of CFD in ship hydrodynamics is such that the steady resistance and flow for the zero-drift case are almost as accurate as the experimental data, although certain issues need further improvements.¹³ An extension of these methods to simulate unsteady ship motion has also been initiated.¹⁴⁻¹⁶

Nevertheless, relatively little attention has been given to an extension of the RaNS equation methods in order to consider free-surface effects in the drift condition. This may be due to the difficulties of simulating the complicated free-surface flow features, e.g., those mentioned above. They involve a considerably higher computational load than that required for the zero-drift condition and, more importantly, there is limited information from experimental fluid dynamics (EFD) to allow a detailed evaluation and validation of the CFD results. The most detailed recent EFD work on this problem has been by the authors;³ however, only a few CFD results were presented for comparison with the same EFD condition.^{17,18}

In this study, a large-domain approach¹⁹ is applied to the problem, with extensions for application to the steady-drift condition. The RaNS and continuity equations are solved with the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/ free-surface conforming grid. The results are evaluated through comparisons with recently completed extensive EFD results for the Series 60 $C_{\rm B}$ = 0.6 ship model at low (0.160) and high (0.316) Fr for drift angle (β) = 10°.³ The former case essentially simulates the zero-Fr condition, so that the comparisons with the latter case allow the identification of salient features of the wave-induced effects. In the earlier work,¹⁹ results for zero drift angle and similar comparisons were presented for the same geometry with detailed experimental data.^{20,21} Owing to the limitations of a numerical treatment of the free-surface boundary conditions, breaking-wave and bubble entrainment effects will not be discussed in detail here.

This paper gives an overview of the present numerical approach, the computational conditions, and numerical uncertainty analysis. The results are presented for wave and flow fields, with an emphasis on important flow features of drift- and wave-induced effects and comparisons with experiments. Finally, conclusions from the present study are given, with recommendations for future work.

Computational method

The computational method is basically the largedomain approach¹⁹ for the steady, straight-ahead condition. Some extensions are made to include both sides of the hull when considering the drift condition. The large-domain approach¹⁹ is based on extensions and modifications of the interactive approach.^{22,23} Additional information on the numerical approach is provided in the literature,²⁴ including a transition for design applications, and extensions for naval combatants with bulbous bows and transom sterns, utilizing multiblock domain decomposition and propeller–hull interactions, as described elsewhere.²⁵ Here, an overview is given.

The numerical method solves the unsteady RaNS and continuity equations for mean-velocity (U, V, W) piezometric pressure (\hat{p}) and eddy viscosity v_t by using the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. It should be noted that a relatively simple twolayer algebraic turbulence model was chosen in the present study because recent studies have shown that any other isotropic model has the same problem, i.e., a difficulty in correctly reproducing mean flow and turbulence fields associated with strong longitudinal vortices. A better alternative would be a full Reynolds stress model, but that is not the main focus of this work. The equations are transformed from Cartesian coordinates (X, Y, Z) in the physical domain to numerically generated, boundary-fitted, nonorthogonal, curvilinear coordinates (ξ, η, ζ) in the computational domain. A partial transformation is used, i.e., coordinates but not velocity components (U, V, W). The equations are solved using a regular grid, finite-analytic spatial and first-order backward difference temporal discretization, a PISO (Pressure-Implicit with Splitting of Operators)-type pressure algorithm, and the method of lines. For steadyflow applications, time serves as a convergence parameter, and the grid is updated at each time step to conform to both the body and the free surface.

As shown in Fig. 1, we considered a ship fixed in a uniform onset flow $\mathbf{U}_0 = (U_0 \cos \beta, U_0 \sin \beta, 0)$, where β is the drift angle. Take the Cartesian coordinate system with the origin on the undisturbed free surface, the Xand Y axes in the horizontal plane, and the Z axis directed vertically upward. Figure 1 also shows the solution domain. For application to the present drift condition, the solution domain includes both port and starboard sides of the hull. As shown in Fig. 1, the specified boundaries are the body surface S_b , the inlet plane S_i , the exit plane S_e , the outer boundary S_o , and the free-surface S_{ζ} , which for zero Fr becomes the symmetric water plane S_w . The boundary conditions for zero Fr are as follows: on Sb, $(U, V, W) = \partial \hat{p}/\partial n = 0$ (where n Y. Tahara et al.: Steady drift motion



Fig. 1. Definition sketch of the coordinate system, solution domain, and boundaries

indicates normal to the body); on S_i , freestream values are imposed, i.e., $U = U_0 \cos \beta$, $V = U_0 \sin \beta$, $W = \hat{p} = 0$; on $S_{\rm e}$, the axial diffusion and pressure gradient are assumed to be negligible, i.e., $\partial^2(U, V, W)/\partial X^2 = \partial \hat{p}/\partial X = 0$; on S_w , $\partial (U, V, \hat{p}) / \partial Z = W = 0$; on $S_0, U = U_0 \cos \beta, V = U_0 \sin \beta$, $W = \partial_n / \partial n = 0$ (where n is normal to the surface). For nonzero Fr, the boundary conditions are similar except on S_{ζ} , where exact nonlinear kinematic and approximated dynamic free-surface conditions are applied on the exact free surface, which is determined as part of the solution; i.e., the dynamic conditions $\partial(U, V, W)/\partial Z = 0$ and $\hat{p} = \zeta/Fr^2$ are applied to the velocity and the pressure, with ζ determined through the solution of the exact nonlinear kinematic condition using a Beam and Warming linear multistep scheme with both explicit and implicit 4th-order artificial dissipation and local timestepping based on the local velocity magnitude. The boundary conditions for ζ are: on S_i , $\zeta = 0$; on S_o , $\partial \zeta / \partial n$ = 0; on $S_{\rm e}$, $\partial \zeta / \partial X = 0$.

The RaNS grid is an H-type, with constant-X planes stacked to form a complete three-dimensional grid, which appeared to be suitable for application to the fine ship considered in the present study. The bow and stern are resolved by axial clustering of grid points distributed according to hyperbolic tangent stretching functions. Because of the H-type grid, nonvertical sterns are resolved in a staircase fashion. The constant-X crossplane grids are generated elliptically by solving a Poisson equation for the transformation between (Y, Z)and (η, ζ) . Spacings are specified in the η -direction on the surface of the hull, which for the Baldwin-Lomax turbulence model should be at a $Y^+ \approx 1$, and in the ζ -direction at both the centerplane and the free surface. The initial grid must extend to an elevation sufficiently above the zero, or design waterline to allow for wave crests. As the wave field develops, the RaNS grid conforms to the free surface. By conforming the grid points and saving the initial distribution along $\eta =$

constant, or girth-wise lines, the grid is easily updated; the point on the free surface moves to a new elevation, and all points below the free surface slide along the η = constant line so as to maintain their initial relative distribution.

The kinematic free-surface boundary condition grid, which is independently generated from the RaNS grid, is two-dimensional (i.e., a function of (X, Y)), is updated iteratively to fit the wave-hull intersection, and is different from the RaNS grid in that, instead of high near-wall resolution, more points are distributed in the outer flow to resolve the wave field. The grid is 460×200 to include both sides of the hull, and consists of equal spacing in the axial direction and a power distribution in the transverse direction. Communication between the RaNS and free-surface grid is accomplished using bilinear interpolation such that the velocity field from the $\zeta = 1$ and kp1 (minimum and maximum indices of grids in the ζ direction, corresponding to port- and starboard-side free surfaces, respectively) plane is interpolated to the free-surface grid. Similarly, the wave elevation is interpolated from the free-surface grid to the $\zeta = 1$ and kp1 plane of the RaNS grid. More details of this numerical method are given in the literature.^{19,24}

Experimental data and computational conditions and grids

Recently, extensive EFD results were obtained at the Iowa Institute of Hydraulic Research for the Series 60 $C_{\rm B} = 0.6$ ship model in steady drift motion.³ The data include photographs and videos, resistance, side force, and yaw moments, sinkage, trim, and heel angles, wave profiles along the hull and wave elevations, and mean velocity and pressure fields for numerous crossplanes from the bow to the near wake. In this study, detailed descriptions are provided of the experimental equipment, procedures, and uncertainty analysis. The Series 60 $C_{\rm B} = 0.6$ ship model is a single-propeller merchant-type ship which is a standard for ship-hydrodynamics research, and was particularly chosen with three other hull types as a representative hull form for the Cooperative Experimental Program (CEP).²⁶

The conditions basically simulate the experiments, i.e., $U_o = 1$, L = 1, and for both zero ($\beta = 0^\circ$) and nonzero ($\beta = 10^\circ$) drift angles, for low Fr, Fr = 0 and Re = 2×10^6 , and for high Fr, Fr = 0.316 and Re = 4×10^6 . For $\beta = 10^\circ$, averaged experimental Re values for mean flow measurements are about 30% higher than those of the computational conditions. However, the influence of these 30% differences in Re on major aspects of mean flow field were shown to be small in the present computation. Note that the experimental low-Fr case is defined as Fr = 0.16; however, free-surface effects are negligible



Fig. 2. Computational grid. **a** Overview. **b** Top view. **c** Cross section, X = 0.1, Fr = 0.316

except near the bow. A partial view of the Fr = 0.316 RaNS grid is shown in Fig. 2. For both Fr, the inlet, the exit, and the outer boundaries are located at X = (-0.4, 2.0) and r = 1, respectively. The first grid points off the body surface are located in the range $Y^+ < 2(= \text{Re } U_\tau Y_n)$. The RaNS grids are prepared as follows: for Fr = 0, 90 × $40 \times 60 = 216\,000$ (Grid A), $90 \times 60 \times 60 = 324\,000$ (Grid Y. Tahara et al.: Steady drift motion

B), $90 \times 80 \times 60 = 432\,000$ (Grid C), and $108 \times 48 \times 72 =$ 373248 (Grid D) (in longitudinal \times radial \times girthwise directions); and for Fr = 0.316, $160 \times 60 \times 71 = 681600$ (Grid E), $180 \times 60 \times 80 = 864000$ (Grid F), and 180×80 \times 80 = 1152000 (Grid G). The free-surface boundary condition grid size is $460 \times 100 = 92000$ for both portand starboard-side free surfaces for all Fr = 0.316 grids. The values of the time increment and underrelaxation factors for velocity and pressure are 0.01, 1, and 0.1 for Fr = 0, and 0.01, 1, and 0.01 for Fr = 0.316, respectively. The computations are completed for 40 and 80 nondimensional times for Fr = 0 and 0.316, respectively, although the convergence criterion is satisfied earlier, so that the residual defined by Eq. 31 in Tahara and Stern¹⁹ for all variables will be about 10⁻⁴, which is satisfied in 20 and 40 nondimensional times for Fr = 0 and 0.316, respectively. The computed hydrodynamic forces indicate small oscillations with a minimal reduction in the residuals after the above-mentioned convergence criterion is satisfied. This may be due to iterative nonconvergence, or possibly to the unsteady nature of the flow involving three-dimensional flow separation associated with unsteady vortex shedding. The oscillation of the solutions is considered in the analysis of iteration uncertainty described in the following section.

Uncertainty analysis

Some more detailed characteristics of the present computational grids are now discussed in conjunction with uncertainty analysis. The numerical uncertainty analysis should include iterative and grid-convergence tests, and possibly order-of-accuracy studies. Note that the derivation of the finite-analytic method precludes term-byterm error analysis, and that equation error depends on both cell Re and aspect ratio. Order-of-accuracy studies on the present finite-analytic method has been carried out^{24,27} for both simple and practical geometries, and it was stated that orders-of-accuracy vary between about 1.5 and 2.5 depending on geometry and flow complexity. The verification procedure for the present results basically follows that of Stern et al.,24 and validation processes described by Coleman and Stern²⁸ are also needed.

Coleman and Stern²⁸ defined comparison error as the resultant of all errors associated with the data and all errors associated with the simulation, and this is written as the difference E = D - S. The uncertainty, $U_{\rm E}$, in the comparison error is defined as $U_E^2 = U_D^2 + U_S^2$, where U_D and $U_{\rm S}$ are experimental and simulation uncertainties, respectively. $U_{\rm S}$ can be written as $U_{\rm S}^2 = U_{\rm SN}^2 + U_{\rm SPD}^2 + U_{\rm SMA}^2$, where $U_{\rm SN}$ is simulation numerical uncertainty, $U_{\rm SPD}$ is uncertainty based on a sensitivity analysis of the model coefficients, and $U_{\rm SMA}$ is a model assumption.

Hence, the validation uncertainty $U_{\rm V}$, defined as $U_{\rm V}^2$ = $U_{\rm E}^2 - U_{\rm SMA}^2 = U_{\rm D}^2 + U_{\rm SN}^2 + U_{\rm SPD}^2$, is key metric in the validation process. In the present study, $U_{\rm SPD}$ is ignored, and $U_{\rm SN}$ in Coleman and Stern²⁸ is evaluated by $U_{\rm SN}^2$ = $U_{\rm SG}^2 + U_{\rm SI}^2$, where $U_{\rm SG}^2 = 3\varepsilon/(r^p - 1)$, $\varepsilon = (\phi_1 - \phi_2)/\phi_1$ in percent or for a relative expression, ϕ_1 and ϕ_2 are variables or integral values for finer and coarser grids, respectively, r is a grid refinement ratio, and p is the order of accuracy. $U_{\rm SI}$ is the iterative convergence uncertainty, whose estimation is based on an evaluation of the iteration records of integral and point variables. The level of the iterative convergence is determined by the number of orders of magnitude reduction and magnitude in the residuals $\zeta = \phi^n - \phi^{n-1}$, where *n* is the iteration number and ϕ can be either the solution variables or equation imbalances obtained by back-substitution. For simple geometries and flows, 16 orders of magnitude reduction of ζ to machine zero is possible, so that the iterative convergence uncertainty is negligible. However, for practical geometries and flows, only a few orders of magnitude reduction in ζ to about 10⁻⁴ may actually be attainable. In this case, the estimates for iterative convergence uncertainty are based on statistics of the iteration records of integral and point variables, and are taken as roughly one-half of the difference between the maximum and minimum values.

Table 1 shows the forces and moment of the present results for Fr = 0 and 0.316, and the grid convergence is given in Table 2. The above-mentioned numerical uncertainties are shown in Table 3, including the resultant validation uncertainty $U_{\rm V}$, where $U_{\rm D}$ is given by EFD results in Longo and Stern.³ In the analysis of U_{SG} , an assumed value of p = 1.5-2.5 is used, ε is evaluated between grids (A) and (D), and (E) and (G) are used for Fr = 0 and 0.316, respectively. U_{st} is evaluated based on one-half of the difference between the maximum and minimum values of the iteration records for the last 10 nondimensional times, for which solutions for grids (D) and (G) are used for Fr = 0 and 0.316, respectively. Note that as the number of total grid points increases, then $U_{\rm SI}$ tends to be larger. The numerical uncertainties in the velocity fields are evaluated at several points in the wake region of the propeller disk, and the averaged values are presented.

In summary, for Fr = 0, the evaluated numerical uncertainties U_{SN} are about 1%–5% for hydrodynamic forces and 1%–4% for mean-flow solutions, and the validation uncertainties U_V for mean flow are about 2%–4%. On the other hand, the values of U_{SN} for Fr = 0.316 are about 1%–4% for hydrodynamic forces, about 2%–4% for wave profiles and wave elevations, and about 1%–4% for mean-flow solutions. The U_V values for Fr = 0.316 are about 4% for wave profiles and wave elevations, and about 2%–4% for mean-flow solutions.

Table 1. Lift, drag, and moment							
		Fr =	$=0, \beta = 10^{\circ}$			Fr = 0.316, $\beta = 10^{\circ}$	
Conditions	$(A) \\ 90 \times 40 \times 60$	$(B) 90 \times 60 \times 60$	(C) $90 \times 80 \times 60$	$(D) 108 \times 48 \times 72$	$(E) (E) 160 \times 60 \times 71$	$(F) \\ 180 \times 60 \times 80$	$(G) \\ 180 \times 80 \times 80$
$ift (CL = L/0.5\rho S_{DWL} U_0^2)$	1.826E-02	1.837E-02	1.836E-02	1.792E-02	1.868E-02	1.891E-02	1.898E-02
(%CL) hydrostatic (z/Fr^2)	0.00	0.00	0.00	0.00	-23.46	-23.46	-23.46
$(\% CL)$ piezometric (\hat{p})	102.25	102.19	102.21	102.21	125.56	125.48	125.45
(%CL) frictional	-2.25	-2.19	-2.21	-2.21	-2.10	-2.02	-1.99
Drag (CD = $D/0.5\rho S_{\text{DWI}} U_o^2$)	8.445E-03	8.406E-03	8.381E-03	8.298E-03	1.030E-02	1.047E-02	1.046E - 02
($\%$ CD) hydrostatic (\ddot{z}/Fr^{2})	0.00	0.00	0.00	0.00	-7.53	-7.53	-7.53
$(\% CD)$ piezometric (\hat{p})	51.50	52.40	52.40	50.40	71.08	71.94	72.47
(%CD) frictional	48.50	47.60	47.60	49.60	36.45	35.59	35.06
Aoment (CMz = Mz/0.5 $\rho L_{\rm pp}^3 U_{\rm o}^2$)	-1.186E-03	-1.188E - 03	-1.190E-03	-1.174E-03	-1.394E-03	-1.408E-03	-1.408E-03

		Fr =	$0, \beta = 10^{\circ}$			Fr = 0.316, $\beta = 10^{\circ}$	
Conditions	$(A) \\ 90 \times 40 \times 60$	$(B) 90 \times 60 \times 60$	$(C) 90 \times 80 \times 60$	$(D) 108 \times 48 \times 72$	(E) $160 \times 60 \times 71$	$(F) (E) \times 60 \times 80$	$(G) \\ 180 \times 80 \times 80$
ε : Lift (CL = $L/0.5\rho S_{\text{nwr}} U_2^2$)		0.58%	-0.01%	-1.91%		1.19%	1.58%
ε : Drag (CD = $D/0.5\rho S_{\text{DWI}}^{2}U_{0}^{2}$)		-0.47%	-0.30%	-1.78%		1.61%	1.54%
ε : Moment (CMz = Mz/0.5 $\rho L_{nn}^{3}U_{0}^{2}$)		0.17%	0.11%	-1.01%		0.96%	0.97%
Evaluation of ε		Bet.(A)–(B)	Bet.(B)–(C)	Bet.(A)–(D)		Bet.(E)–(F)	Bet.(E)–(G)

Table 2. Grid convergence

Results

Here, the results for Fr = 0 and 0.316 are presented and discussed, including a comparison with experimental data. In general, the emphasis of the discussions concerns the comparisons and evaluation of the computational method, although reference is made where appropriate Longo and Stern³ for a relevant discussion and interpretation of the flow physics. However, in some cases, certain discussions are replicated for clarity of presentation. The discussions include a comparison between zero ($\beta = 0^{\circ}$) and nonzero ($\beta = 10^{\circ}$) drift angle results, although in some cases, the former are not shown in the present figures. See Tahara and Stern¹⁹ for details of $\beta = 0^{\circ}$ solutions, including comparisons with the experimental data of Toda et al.²⁰ and Longo et al.²¹

Integral variables

First, integral variables are discussed. Table 1 shows a comparison of the forces and moments of the present results. It should be noted that the sinkage and trim are all fixed in the present computation but free in the measurements, which precludes a quantitative comparison between the two sets of results. Therefore, the comparison is focused on the trend between high and low Fr. As indicated in Table 1, the differences between Fr = 0.316 (grid (G)) and Fr = 0 (grid (D)) for CL, CD, and CMz are about -6%, -21%, and -17% in magnitude, and those for experiments between high (0.316) and low (0.16) Fr are about -16%, -26%, and -39%, i.e., the same trends are predicted by the present method. The differences between the present data and the experimental data can likely be attributed to the abovementioned differences in the fix/free condition. In addition, some shortcomings of the present method in resolving complicated features in the flow field may be related to the differences, e.g., the absence of wavebreaking and bubble-entrainment effects, as well as the underpredicted magnitude of axial vortices. These are discussed later in conjunction with a comparison of the computational results with the experimental data.

Surface pressure and frictional streamlines

Next, surface-pressure distributions are considered. The surface-pressure and axial and vertical surface-pressure gradient contours for $\beta = 10^{\circ}$ and Fr = 0 and 0.316 are shown in Fig. 3. Unfortunately, no experimental data are available for comparison. Significant differences are observed between the present and some precursory $\beta = 0^{\circ}$ results.¹⁹ The differences are mainly due to increases and decreases in pressure on the port and starboard sides, respectively, which correspond to the pressure and suction sides, respectively. In addition, differences

		Fr	$=0, \beta = 10^{\circ}$	>			Fr =	$0.316, \beta = 1$.0°	
Conditions	$\overline{U_{ m SG}\left(\% ight)}$	$U_{\mathrm{SI}}\left(\% ight)$	$U_{ m SN}$ (%)	$U_{ m D}$ (%)	$U_{ m v}\left(\% ight)$	$U_{ m SG}\left(\% ight)$	$U_{\mathrm{SI}}\left(\% ight)$	$U_{ m SN}\left(\% ight)$	$U_{ m D}$ (%)	$U_{\mathrm{V}}\left(\% ight)$
Lift (CL)	2.1-4.9	0.4	2.2–4.9	_	_	1.7–3.9	0.3	1.7–3.9	_	_
Drag (CD)	2.0-4.6	0.2	2.0-4.6			1.7-3.8	0.5	1.7-3.8		—
Moment (CMz)	1.0-2.4	0.4	1.1-2.4			1.1-2.4	0.3	1.1-2.4		—
Wave profiles			_	2.6		1.8 - 4.0	0.6	1.9-4.0	1.3	2.3-4.2
Wave elevations			_	2.2		1.7 - 3.9	0.6	1.8-3.9	1.1	2.1-4.1
Mean flow										
$U(\% U_{0})$	1.8 - 4.0	0.7	1.9-4.0	1.5	2.4-4.3	1.8-4.1	0.8	2.0-4.2	1.5	2.5-4.4
$V(\% U_{0})$	1.3-3.0	0.7	1.5-3.1	1.5	2.1 - 3.4	1.4-3.2	0.8	1.6-3.3	1.5	2.2-3.6
$W(\% U_{0})$	1.3-3.0	0.7	1.5-3.1	1.5	2.1 - 3.4	1.4-3.2	0.8	1.6-3.3	1.5	2.2-3.6
$C_{\rm p}$	0.4 - 1.0	0.4	0.6–1.1	3.0	3.1-3.2	0.5 - 1.2	0.6	0.8–1.3	3.0	3.1–3.3

Table 3. Summary of numerical and experimental uncertainties

-, not applicable or negligible

between Fr = 0 and 0.316 are clearly displayed, which are due to the nonlinear free-surface effects for the latter case. Figure 4 shows computed frictional streamlines for $\beta = 10^{\circ}$, and Fr = 0 and 0.316. For Fr = 0.316, flow patterns correlate with the pressure and pressure gradients shown in Fig. 3. On the forebody port side, a region of low pressure is observed near the bilge, which causes the flow to converge towards the keel, partially converge at the keel, and separate to generate a forebody-keel vortex. On the forebody starboard side, the streamlines are downward and towards the bilge, where they meet the streamlines from the forebody port side, and finally the streamlines form an open-type three-dimensional separation pattern that is related to the generation of a forebody-bilge vortex. A closedtype separation region is also seen near the bow on the starboard side. On the other hand, on the afterbody starboard side, a low pressure region is located near the stern bilge which causes the streamlines to converge towards the stern bilge, meet the streamlines from the port side, and finally form an open-type separation pattern, which results in the generation of an afterbodybilge vortex. Streamlines on the afterbody port side partially converge at the keel and then separate to generate an afterbody-keel vortex. Similar aspects of frictional streamlines are seen for Fr = 0, but in this case, free-surface effects are absent and the differences are obvious, especially near the waterline. In addition, for Fr = 0, a larger region of closed-type separation is observed near the bow on the starboard side than for Fr = 0.316.

Wave profiles

The wave profile at the hull is compared with the experimental data in Fig. 5. The results for both port and starboard sides show fairly close agreement with the measurements with respect to wave amplitude, shape, and phase, although some systematic differences can be seen. The amplitudes of the port-side bow-wave crest and trough are somewhat underpredicted. Similarly, on the starboard side, the wave amplitudes are slightly underpredicted on the forebody. Some underprediction near the bow on the starboard side is likely related to the limitations of the present numerical method, i.e., although the experiments showed flow separation associated with wave-breaking in this region, that is not completely reproduced in the present method. For both sides of the hull, good agreement is observed after X =0.5, although some details on the starboard side around X = 0.9 were not completely predicted.

The trends and differences between $\beta = 0^{\circ}$ and $\beta = 10^{\circ}$ are mostly reproduced in the present computation, i.e., the greatest changes to the profiles ($\Delta \zeta$) occur upstream of x = 0.25. For x < 0.25 the port-side profile shifts upward significantly with increases in β , and the largest increase in profiles occur where $\zeta_x = 0$. The starboardside profile decreases significantly with increases in β , and again the largest $\Delta \zeta$ occurs where $\zeta_x = 0$. For x >0.25, the influence of β is fairly small except for decreases and increases in the fore and aft shoulder-wave troughs, respectively.

Wave elevations

Figure 6 shows the contours of the wave elevations and axial wave slopes for Fr = 0.316, and $\beta = 0^{\circ 19}$ and 10° . Similar trends and differences between $\beta = 0^{\circ}$ and 10° are shown between the present results and the measurements, i.e., the wave amplitudes clearly increase on the port side and decrease on the starboard side of the model with increasing β , and for $\beta = 10^{\circ}$, as with $\beta = 0^{\circ}$, ζ_x is in phase with ζ and has similar patterns, but the magnitudes are significantly increased and decreased globally on the port and starboard sides, respectively. For $\beta = 10^{\circ}$, the wave patterns become asymmetrical, and the computed elevations show close similarities with the experimental data with regard to both amplitude



Fig. 3. Computed surface-pressure and axial and vertical surface-pressure gradient contours for $\beta = 10^{\circ}$

Y. Tahara et al.: Steady drift motion



Fig. 5. Comparison of wave profiles for Fr = 0.316 and $\beta = 10^{\circ}$. a Port (pressure) side. b Starboard (suction) side

and shape for forebody, i.e., experimental ζ_x values indicate that the port bow-wave crestline curves back towards the model with increasing X, and a similar trend is observed in the present results. However, in the global region on the starboard side, the bow wave system propagating downstream is not very clearly reproduced. The results for the afterbody also show fairly close similarities, but the detailed resolution, particularly of the complex global-region wave system on the starboard side, is incomplete. The differences between the present and the experimental results increase as the distance from the hull increases, which is likely due to the decrease in computational grid density in that region.

Mean velocity and pressure field

Finally, the results for the mean velocity and the pressure field are shown in Figs. 7–10. Figures 7 and 8, and 9

and 10 show the results for Fr = 0 and 0.316, respectively, and Figs. 8 and 10 are from previous experimental results.^{3,29} The contours of ω_x , H, and C_p are shown at the same crossplanes as those of the measurements, i.e., X = (0, 0.1, 0.2, 0.4, 0.6, 0.8, 0.9, 1.0, 1.1, 1.2). For both the present and the experimental results, the overall trends are similar between the Fr = 0 and 0.316 flow fields, in which significant free-surface effects are also clear for the latter. Most of the flow features in the drift condition are significantly different from those for the zero-drift case. In many respects, the flow is completely altered. The boundary-layer and wake development is dominated by strong crossflow effects and vortices, as opposed to the axial pressure gradients and weak crossflow effects observed in the $\beta = 0^{\circ}$ case. The waveinduced effects at Fr = 0.316 are similarly explained as for $\beta = 0^{\circ}$, i.e., the Fr-related differences in the velocity and pressure correlate with the wave field, which,



Fig. 6. Computed and measured wave-elevation and wave-slope contours for Fr = 0.316. a Computations. b Measurements
Y. Tahara et al.: Steady drift motion





Fig. 8. Mean-flow measurements for $\beta = 10^{\circ}$ and Fr = 0.16. **a** Axial vorticity. **b** Total head. **c** Pressure

Fig. 7. Mean-flow solutions for $\beta = 10^{\circ}$ and Fr = 0. **a** Axial vorticity. **b** Total head. **c** Pressure

however, is significantly more complex for $\beta = 10^{\circ}$ than for $\beta = 0^{\circ}$, creating a more complex boundary-layer and wake response. More details are discussed below.

Figures 9a and 10a show the axial vorticity contours for Fr = 0.316. For both experimental and computational results, the extensive vorticity $(\omega_{\rm r})$ in the flow field is evident. Note that the contour ranges for the present and the experimental results are different, i.e., $-20 < \omega_r < 20$ and $-50 < \omega_r < 50$ for the former and latter, respectively. In the present results, the magnitudes of ω_r are generally underpredicted, although many important aspects of the vortices near the hull are reproduced. On the forebody, the keel and bilge vortices are visible beginning at X = 0.1, where the keel vortex is relatively weak, and are not evident beyond X = 0.4, and the bilge vortex is relatively strong. The vortex core is off the body and moves further from the centerplane with increasing X. On the afterbody, the forebody bilge vortex weakens, and an afterbody bilge vortex develops as with $\beta = 0^{\circ}$, where the vorticity has a core region that is off the body and toward the free surface, with a tail that extends toward the centerplane, and there appears to be a weak interaction with the forebody bilge vortex. In the present results, the forebody bilge vortex weakens faster than that for measurements, which is likely due to the lower accuracy of the turbulence model and the longitudinal grid resolution. In the region near the stern, the present results show several few similarities with the measurements, e.g., a counter-rotating keel vortex is evident at X = 1 (AP), although the magnitude is underpredicted. The important flow aspects in the wake region are also similar between the present and the experimental results. The forebody bilge vortex dissipates and is diffuse in its trajectory towards the free surface, where the afterbody bilge vortex becomes oval-shaped and dissipates in its trajectory off the body towards the free surface, the afterbody keel vortex dissipates relatively fast in its trajectory, as does the afterbody bilge vortex, and there is limited interaction between fore and afterbody bilge vortices. In the experimental data, a wave-induced vortex is evident on the port-side forebody which initiates between 0.2 < X <0.4 underneath the breaking bow wave, and follows a trajectory near the free surface along the side of the



Y. Tahara et al.: Steady drift motion



Fig. 10. Mean-flow measurements for $\beta = 10^{\circ}$ and Fr = 0.316. a Axial vorticity. b Total head. c Pressure

Fig. 9. Mean-flow solutions for $\beta = 10^{\circ}$ and Fr = 0.316. **a** Axial vorticity. **b** Total head. **c** Pressure

hull. Although the present free-surface boundary conditions do not include exact breaking-wave effects, a similar wave-induced vortex is reproduced in the region, but the magnitude is generally reduced.

Figures 7a and 8a show the axial vorticity contours for low Fr, i.e., Fr = 0 and 0.16 for the present and experimental results, respectively. In the measurements, the overall flow pattern is similar to that for Fr = 0.316, but with two important differences: the bilge and keel vortices appear weaker and the trajectories are altered somewhat, and owing to the reduced wave field, there is no wave-induced vorticity in the flow field. The similar trend in the differences between low and high Fr are displayed in the present result, but again the shortcomings discussed for Fr = 0.316 hold true for Fr = 0. In addition, some wave-induced effects on the flow field are observed in the measurements, especially in the region near the bow on the port side, although the influences are much smaller those for Fr = 0.316.

In Figs. 7b–10b, the total head (H) field is presented, where the viscous regions (H) on the hull and in the

wake in the present results and the measurements are compared. In the experimental data for Fr = 0.316, the patterns correlate with ω_x and the boundary layer and wake losses, but with stronger interactions than for low Fr between the loss regions of the keel and bilge vortices on the afterbody, creating a somewhat more complex pattern. Several important flow aspects observed in the measurements are reproduced in the present results, although some details in the near wake are not complete. For both results, *H* displays similar Fr differences as ω_x for the wave-induced vortex, i.e., for Fr = 0.160, in contrast to ω_x which decreased in magnitude with decreasing Fr, *H* is somewhat increased in magnitude. This is a Reynolds number effect, i.e., the viscous regions are thicker for lower Fr.

Lastly, the pressure field is considered. In Figs. 7c– 10c, the present and measured pressure (C_p) fields around the hull are displayed. In general, the present results show many similarities with the measurements. At the forebody for Fr = 0.316, C_p correlates with U in such a way that the trends are the same but the magnitudes are reversed, especially in the bow and stern regions and at the midbody, where the flow accelerates and the pressure is low. An asymmetric stagnation-type flow is exhibited at the FP. Generally, high- and lowpressure regions exist on the port and starboard sides, respectively, the lowest pressures are in regions of high $\omega_{\rm r}$ with minimums in the core regions, and the bow wave stagnation effects are evident as increased pressures at X = 0 and 0.1. In both results for Fr = 0.316, the pressure differences (ΔC_p) between port and starboard sides are reduced near the midbody at X = 0.4, and at the afterbody there is a continued reduction in $\Delta_{\!p}$ up to X = 0.9, when $\Delta C_{\rm p}$ increases with the lowest $C_{\rm p}$ in regions of high ω_x and minimums in the core regions, i.e., for the forebody bilge vortex and afterbody bilge vortex. In both results, it appears that most of the sideforce is apparently generated near the bow and fairly near the stern, where ΔC_{p} is largest. For Fr = 0, the present results indicate that the general patterns are similar to those for Fr = 0.316 except for some diminished features due to Fr and viscous effects, i.e., the pressures at the bow and stern are lower owing to the reduced port-side bow wave system and wave effects at the AP and wake, respectively, and the pressure field is higher over the midbody and stern, especially in the regions of the vortex cores. Similar trends are observed in the experimental data. As described above, many important features in the measurements are simulated in the present method. However, some details related to the magnitude of longitudinal vortices are not complete, which can be attributed to the previously mentioned shortcomings, i.e., an inadequate turbulence model and grid resolution.

Summary and conclusions

This paper has shown a comparison of CFD and EFD for boundary layers and wakes and wave fields around the practical 3-D geometry of the Series 60 $C_{\rm B} = 0.6$ ship model in steady drift motion. The numerical method solves the unsteady RaNS and continuity equations with the Baldwin-Lomax turbulence model, exact nonlinear kinematic and approximate dynamic free-surface boundary conditions, and a body/free-surface conforming grid. EFD results from towing-tank experiments at both low (0.16) and high (0.316) Froude numbers are used for the comparison, in which the former case essentially simulates the zero Froude number condition so that comparisons with the latter case allow an identification of the salient features of the waveinduced effects. In addition, a comparison of the results with those from an earlier study for the straight-ahead condition allows an investigation of drift- and waveinduced effects.

In the present calculations, many important flow features displayed in experiments are reproduced, i.e., the asymmetric wave field close to the hull, mean-flow fields dominated by strong crossflow effects that drive the

flow from the port to the starboard side, and asymmetric vorticity developments at the forebody bilge, forebody keel, afterbody bilge, and afterbody keel are correctly simulated. In addition, trends between low and high Fr for integral variables and mean-flow fields show good agreement with the measurements. However, complex details regarding vortex generation and evolution are underestimated in magnitude, amplitudes in the globalregion wave field are underpredicted, and the bow-wave system for the starboard side are not completely reproduced in the global region. The issues for the wave fields are similar to those in the previous work on zero-drift conditions,19 which may be because there was no particular improvement in the numerical techniques in this study. The same holds true for the issues regarding the resolution of vortices, i.e., the turbulence model and the numerical scheme are same as those of Tahara and Stern,¹⁹ and the longitudinal grid density is also similar to that used previously.¹⁹ Although the results obtained by the present method are very promising, further improvements must be made in order to solve these problems. In addition, the inclusion of breaking-wave and bubble entrainment effects are important in order to simulate the flow in the drift condition more accurately.

Acknowledgments. This research was sponsored by the Office of Naval Research under Contract N00014-96-0018, including an allocation for supercomputing hours at DOD HPC resources (NAVO T90), both of which are under the administration of Dr. E.P. Rood, whose support is greatly appreciated. This research was also supported by the Japan–US Cooperative Science Program of the National Foundation of Science of the USA under Contract INT-9513138 and the Japan Society for the Promotion of Science. The authors' thanks are extended to everyone concerned in the program for their kind support and encouragement.

References

- Day WG, Hurwitz RB (1980) Propeller-disk wake survey data for model 4989 representing the FF 1055-class ship in a turn and with a bass dynamometer boat. Report no. SPD-0011-21, David Taylor Naval Research and Development Center, Bethesda, MD
- 2. Longo J, Stern F (1995) Evaluation of surface–ship resistance and a propulsion model-scale database for CFD validation. J Ship Res 40:112–116
- Longo J, Stern F (1996) Yaw effects on model-scale ship flow. Proceedings of the 21st Symposium on Naval Hydrodynamics, Trondheim, pp 312–327 (unpublished)
- Tahara Y (1995) Computation of boundary-layer and wake Flows around an IACC sailing yacht. J Kansai Soc Nav Archit Jpn 224:1–11
- Tahara Y (1996) A multi-domain method for calculating boundary-layer and wake flows around an IACC sailing yacht. J Kansai Soc Nav Archit Jpn 226:63–76
- Ohmori T, Miyata H (1993) Oblique tow simulation by a finitevolume method. J Soc Nav Archit Jpn 173:27–34

- Ohmori T, Fujino M, Miyata H, et al. (1994) A study on the flow field around full ship forms in maneuvering motion. 1st Report. In oblique tow. J Soc Nav Archit Jpn 176:241–250
- Ohmori T, Fujino M, Tatsumi K, et al. (1996) A study on the flow field around full ship forms in maneuvering motion. 3rd Report. Flow field around ship's hull in a steady turning condition. J Soc Nav Archit Jpn 179:125–138
- Akimoto H, Hiroshima F, Miyata H (1996) Hydrodynamic design of a sailing boat by a CAD/CFD system with moving coordinates. Proceedings of the 3rd Korea–Japan Joint Workshop on Ship and Marine Hydrodynamics, Taejon, Korea (unpublished)
- Ohmori T, Fjino M, Miyata H (1998) A study on the flow field around full ship forms in maneuvering motion. J Mar Sci Technol 3:22–29
- Hochbaum AC (1998) Computation of the turbulent flow around a ship model in a steady turn and in steady oblique motion. Proceedings of the 22nd Symposium on Naval Hydrodynamics, Washington, DC, pp 550–567 (unpublished)
- 12. Campana EF, Di Mascio A, Penna R (1998) CFD analysis of the flow past a ship in steady drift motion. Proceedings of the 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, pp 151–159 (unpublished)
- 13. CFD Tokyo Workshop (1994) Tokyo (unpublished)
- 14. Rhee SH, Stern F (1998) Unsteady RANS method for surface ship boundary layers and wakes and wave fields. Proceedings of the 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, pp 67–84 (unpublished)
- Wilson R, Paterson E, Stern F (1998) Unsteady RANS simulation of model 5415 in waves. Proceedings of the 22nd Symposium on Naval Hydrodynamics, Washington, DC, pp 532–549 (unpublished)
- 16. Sato Y, Miyata H, Sato T (1999) CFD simulation of 3-dimensional motion of a ship in waves: application to an advancing ship in regular heading waves. J Mar Sci Technol 4:108–116
- Alessandrini BA, Delhommeau G (1998) Viscous free surface flow past a ship in drift and in rotating motion. Proceedings of the 22nd Symposium on Naval Hydrodynamics, Washington, DC, pp 491–507 (unpublished)

- Di Mascio A, Campana EF (1999) The numerical solution of the yaw flow of a free surface. Proceedings of the 7th International Conference on Numerical Ship Hydrodynamics, Nantes (unpublished)
- Tahara Y, Stern F (1996) A large-domain approach for calculating ship boundary layers and wakes for a nonzero Froude number. J Comput Phys 127:398–411
- 20. Toda Y, Stern F, Longo J (1992) Mean-flow measurements in the boundary layer and wake and wave field of a Series 60 $C_{\rm B} = 0.6$ ship model. Part 1. Froude numbers 0.16 and 0.316. J Ship Res 37:360–377
- 21. Longo J, Stern F, Toda Y (1993) Mean-flow measurements in the boundary layer and wake and wave field of a Series 60 $C_{\rm B} = 0.6$ ship model. Part 2. Scale effects on near-field wave patterns and comparisons with inviscid theory. J Ship Res 37:16–24
- Tahara Y, Stern F, Rosen B (1992) An interactive approach for calculating ship boundary layers and wakes for nonzero Froude number. J Comput Phys 98:33–53
- Tahara Y, Stern F (1994) Validation of an interactive approach for calculating ship boundary layers and wakes for nonzero Froude number. J Comput Fluids 23:785–816
- 24. Stern F, Paterson EG, Tahara Y (1995) CFDSHIP-IOWA: computational fluid dynamics method for surface-ship boundary layers, wakes, and wave fields. IIHR Report No. 381, Iowa Institute of Hydraulic Research, Iowa City
- 25. Stern F, Kim HT, Zhang DH, et al. (1994) Computation of viscous flow around propeller–body configurations: Series 60 $C_{\rm B}$ = 0.6 ship model. J Ship Res 38:137–157
- ITTC (1987) Report of the Resistance and Flow Committee. Proceedings of the 18th International Towing Tank Conference, Kobe, Japan, pp 47–92 (unpublished)
- Zhang ZJ, Stern F (1996) Free-surface wave-induced separation. ASME J Fluids Eng 118:546–554
- Coleman HW, Stern F (1997) Uncertainties and CFD code validation. ASME J Fluids Eng 119:795–803
- Longo J (1996) Yaw effects on model-scale ship flows. PhD Thesis, Department of Mechanical Engineering, University of Iowa

-Consideration of Near-Wall Flow Model Including Surface Roughness Effects-

by Yusuke Tahara, *Member* Tokihiro Katsui, *Member* Yoji Himeno, *Member*

Summary

The present study concerns simulation of ship viscous flow at full-scale Reynolds number. The main objectives are two folds: (1) development of Reynolds-averaged Navier-Stokes equation method applicable to full-scale flow simulation: and (2) investigation on appropriate physical model for full-scale Reynolds number in conjunction with consideration of near-wall flow model including surface roughness effects. In particular, the validity and advantage of two-point wall-function approach have been investigated and the extension was made for inclusion of surface roughness effects on flow and resistance; and currently, evaluation for roughness effects on flat plate flow has been completed. The present numerical method for full-scale flow simulation is based on extension of method developed by the present author, such that, in association with standard k- ϵ model, two near-wall models can be employed, i.e., two-layer method and two-point wall-function method; where the latter has been shown more suitable in practical design use.

1. Introduction

Ship designs in next generation will be dramatically different from those currently in use¹⁾. As such, more innovative design concepts will be applied for design of ships with higher performance and overall operation benefit. In the design of such hull forms, much of the current design database, which has been developed over the past 50 years, is not directly applicable. Since it will be prohibitively expensive to quickly expand the design database through model studies, there is strong motivation to develop simulation based design tools, which are able to diminish or eliminate need for model-scale tests

* Osaka Prefecture University

Received 10th July 2002 Read at the Autumn meeting 14, 15th Nov. 2002 and extrapolation of the results to full scale. This is an important background of further development of Computational Fluid Dynamics (CFD); however, many CFD methods, especially for those based on Reynolds-averaged Navier-Stokes (RaNS) equation method, still suffer difficulties in full-scale ship-flow simulation, that Professor V.C. Patel uniquely called "Achilles Heel of CFD²."



-319-

Fig.1 shows Reynolds number (Rn) variation for ships. Towing tank model tests are normally curried out at $Rn \sim O(10^7)$, expecting the results for possible extrapolation to higher order Rn, that is accomplished in aid of the above-mentioned design database. CFD in ship hydrodynamics must cover quite wide range of Rn in the practical application. The Rn limitations of CFD can be attributed to the limitations from those of the numerical algorithm and those inherent in the physical model including near-wall modeling²⁾. In addition, capability to consider surface roughness effects will be essential for accurate estimation of propulsion performance. All of the above-mentioned issues are challenges in next step of CFD development, which leaded to the motivation of the present study.

In terms of numerical aspects of CFD, the above-mentioned Rn limitations of CFD are directly related to the difficulties to maintain accuracy in the resolution of the increasing gradients of velocity and turbulence parameters within a viscous and turbulent layer of decreasing thickness. In use of conventional near-wall turbulence model, no surface roughness effects can be considered, and more importantly, increasing Rn yields finer and finer computational grids near the wall, which easily causes larger skewness and aspect ratio of grids as well as larger computational load. Some numerical schemes simply fail to yield solutions in high Rn. In fact, there are very limited number of reports on development of RaNS equation method for full-scale or high Rn ship-flow simulation³⁾⁻⁷⁾. This is partly due to a fact that experimental work for the scale is also limited^{8),9)}, and unlike the case for model scale, no work has been carried out to establish CFD validation database. In general, these computational and experimental studies on full-scale ship flow have not completed detailed investigation on characteristics of near-wall flow and turbulence parameters, which is necessary for introduction and verification of appropriate near-wall flow model that will clearly reduce the above-mentioned difficulties in high Rn flow simulation.

The present study concerns simulation of ship viscous flow at full-scale Rn in conjunction with consideration of near-wall flow model including

surface roughness effects. The main objectives are two folds: (1) development of RaNS equation method applicable to full-scale flow simulation; and (2) investigation on appropriate physical model for In particular, the validity and full-scale Rn. advantage of two-point wall-function approach³⁾ have been investigated and the extension was made for inclusion of surface roughness effects on flow and resistance; and currently, evaluation for roughness effects on flat plate flow has been completed. In the present paper, first, flat-plate boundary-layer flows are discussed, with emphasis on characteristics of flow and turbulence quantities in the range from model-scale to full-scale Rn, in which for the first time, solutions of two-layer k- ϵ model have been obtained for $Rn=10^6 \sim 10^9$. Next, the present numerical method is applied to full-scale ship flow computation, and the results are compared with estimations from model-scale measurements¹⁰). The present numerical method is based on extension of method developed by the present author¹¹⁾, such that, in association with standard $k \in model$, two near-wall models can be employed, i.e., two-layer method and two-point wall-function method, where the latter has been shown more suitable in practical design use.

2. Computational Method

As the earlier mentioned, a key issue for full-scale ship-flow simulation is to maintain accuracy in the resolution of flow within a viscous and turbulent layer of decreasing thickness. Fig.2 shows how Rn influences grid arrangement for constant minimum distance to the wall, i.e., $y_{j=2}^+ = 0.1$, where $y^+ = RnU_r y$ is dimensionless distance, y is physical distance normalized by a characteristic length, U_r is the friction velocity normalized by the reference velocity, and $\not=1$ and 100 correspond to wall and outer boundaries, respectively. The grid clustering was done by the geometry series expansion, one of the most popular methods for the purpose (upper figure). As Rn increases, minimum physical distance $y_{\min} = y_{j=2}$ (lower figure) and grid density defined by $\Delta y^{-1} = 2/(y^{j+1} - y^{j-1})$ (upper figure) near the wall rapidly decreases and increases, respectively. In addition, the numbers of grid points inside of

-320-

near-wall layer, i.e., $n(y^+ \le 100)$, $n(y^+ \le 500)$, and $n(y^+ \le 1000)$ are also shown in the figure. It is noticed that nearly half of total grids are located in the region very close to the wall. These facts imply that introduction of appropriate modeling of near-wall flow and turbulence parameters will clearly reduce the earlier-mentioned difficulties in high Rn flow simulation. In the following, overview of the present numerical method is given in association with consideration of capability to include surface roughness effects, where a focus is placed on introduction of the two-point wall-function approach.

Consider a ship fixed in the uniform onset flow as depicted in Fig.3. Take the Cartesian coordinate system with the origin on the undisturbed free surface, X and Y axes on the horizontal plane, and Z axis directed vertically upward.



Fig.2 Influences of Reynolds number on grid arrangement for equal dimensionless near-wall distance (Geometry series expansion, $y_{j=2}^+ = 0.1$, $j_{MAX} = 100$).



Fig.3 Definition sketch of coordinate system.

The non- dimensional RaNS equations for unsteady, three- dimensional incompressible flow can be written in Cartesian tensor notation as

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x^j} + \frac{\partial u_i u_j}{\partial x^j} + \frac{\partial p}{\partial x^i} - \frac{1}{Rn} \nabla^2 U_i = 0 \quad (1)$$
$$\frac{\partial U_i}{\partial x^i} = 0 \quad (2)$$

where $U_i=(U, V, W)$ and $u_i=(u, v, w)$ are the Cartesian components of mean and fluctuating velocities, respectively, normalized by the reference velocity U_0 , x=(X, Y, Z) is the dimensionless coordinates normalized by a characteristic length L, p is the pressure normalized by ρU_0^2 , $Rn=U_0L/v_0$ is the Reynolds number, v_0 is the kinematic viscosity, and the barred quantities $-u_i u_j$ are the Reynolds stresses normalized by U_0^2 . If $-u_i u_j$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity v_0 i.e.,

$$-\overline{u_i u_j} = v_t \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i}\right) - \frac{2}{3} \delta_{ij} k \tag{3}$$

where $k=(\overline{uu}+\overline{vv}+\overline{ww})/2$ is the turbulent kinetic energy, equation (1) becomes

$$\frac{\partial U_i}{\partial t} + \left(U_j - \frac{\partial v_i}{\partial x^j} \right) \frac{\partial U_i}{\partial x^j} - \frac{\partial v_i}{\partial x^j} \frac{\partial U_j}{\partial x^i} + \frac{\partial p}{\partial x^i} \left(p + \frac{2}{3} k \right) - \frac{1}{R_{\phi}} \nabla^2 U_i = 0$$
(4)

where $1/R_{\phi} = 1/Rn + v_{\epsilon}$, and $\phi = U_i$ (i=1,2,3). Equations (2) and (4) can be solved for U_i and p when a suitable turbulence model is employed to calculate the eddy-viscosity distribution. In this study, eddy viscosity is given by k and rate of turbulent energy dissipation ε , where two methods are employed to consider near-wall effects, i.e., two-layer method and two-point wall-function method. For both methods,

-321-

eddy viscosity in the outer layer is defined by $v_t = C_{\mu} k^2 \varepsilon$, and k and ε are obtained from the transport equations

$$\frac{\partial k}{\partial t} + \left(U_{j} - \frac{1}{\sigma_{k}} \frac{\partial v_{i}}{\partial x^{j}}\right) \frac{\partial k}{\partial x^{j}} - \frac{1}{R_{k}} \nabla^{2} k - G + \varepsilon = 0$$
(5)
$$\frac{\partial \varepsilon}{\partial t} + \left(U_{j} - \frac{1}{\sigma_{\varepsilon}} \frac{\partial v_{i}}{\partial x^{j}}\right) \frac{\partial \varepsilon}{\partial x^{j}} - \frac{1}{R_{\varepsilon}} \nabla^{2} \varepsilon - C_{\varepsilon 1} \frac{\varepsilon}{k} G + C_{\varepsilon 2} \frac{\varepsilon^{2}}{k} = 0$$
(6)

where R_k and R_{ε} are the effective Reynolds numbers defined by $1/R_{\phi}=1/Rn+\nu_{\sigma}/\sigma_{\phi}$, and $\phi=k, \varepsilon$. Also, G is the rate of production of k, and $(C_{\mu}, C_{\varepsilon l}, C_{\varepsilon 2}, \sigma_k, \sigma_{\sigma})$ are constants whose values are (0.09, 1.44, 1.92, 1.01, 1.3).

For two-layer method, k in the near-wall layer is obtained from the transport equation (5), then v_t and ε are given by $v_t=C_{\mu}k^{1/2}\ell_{\mu}$ and $\varepsilon=k^{3/2}\ell_{\varepsilon}$, where ℓ_{μ} and ℓ_{ε} are the length scales based on distance from the wall y, i.e., $\ell_{\mu}=C_{ly}\{1-\exp(-R_{s'}/A_{\mu})\}$, and $\ell_{\varepsilon}=C_{ly}\{1-\exp(-R_{s'}/A_{\varepsilon})\}$. $R_{y}=Rnk^{1/2}y$ is the turbulent Reynolds number, and $C_{l}=\kappa C_{\mu}$ ^{3/4}, $A_{\mu}=70$, and $A_{\varepsilon}=2C_{l}$ are model constants, where $\kappa=0.418$ is the von Karman constant. The near-wall layer includes the laminar sublayer, the buffer layer, and a part of the fully-turbulent logarithmic layer.

For two-point wall-function method, equations (4) through (6) are solved in the outer layer, and near-wall boundary conditions for U_i , k and ε are given by law of the fully-turbulent logarithmic layer. In use of this method, at least two points (i.e., $\eta=2$ and 3, where $\eta=1$ and 2 are wall surface and boundary of solution domain, respectively) must be located in logarithmic region. The present wall-function is written as

$$\frac{q}{U_{\tau}} = \frac{1}{\kappa} \ln y^{+} + B - \Delta B \tag{7}$$

in which q is the local streamwise velocity, and *B*=5.45 is model constant. ΔB is the so-called roughness characterization function and depends not only on the size but also the type of roughness. White¹²⁾ suggested,

$$\Delta B = \frac{1}{\kappa} \ln(1 + k_s^+) \qquad (8)$$

where $k_s^+ = RnU_rk_s$ and k_s is roughness heights normalized by a characteristic length, by curve fit for uniform sand grain roughness. In the present procedure, a value for U_r is assumed and the boundary conditions at $\eta=2$ are determined from equation (7) and the assumption of local equilibrium for k and ϵ , i.e.,

$$\frac{q_2}{U_r} = \frac{1}{\kappa} \ln y_2^* + B - \Delta B$$

$$k_2 = U_r^2 / \sqrt{C_\mu} , \quad \varepsilon_2 = U_r^3 / \kappa y_2$$
(9)

The numerical solution then provides the velocity at $\eta=3$ and U_r is updated by requiring this velocity also to satisfy equation (7), i.e., by solving

$$\frac{q_3}{U_r} = \frac{1}{\kappa} \ln y_3^+ + B - \Delta B \tag{10}$$

by a root-finding technique. Thus, an iterative procedure is used to satisfy the wall boundary conditions. Note that q_2 and q_3 are local streamwise velocities, hence coordinate transformation is needed to give velocity boundary conditions in the present Cartesian coordinate system.

Finally, the five transport equations (4) through (6) are discretized by the 12-point finite-analytic scheme. The solution of the complete flow equations involves a global iteration process, in which the velocitypressure coupling is effected by predictor-corrector steps, i.e., PISO-type one-step procedure. Reader may refer to reference¹¹⁾ for more details of numerical algorithm and the related references.

3. Flat Plate Flow

Flat-plate turbulent boundary-layer flow has been of classical interest in ship hydrodynamics, since resultant frictional formulas are used to predict total resistance by extrapolation of model results to full scale. In this section, first, results from two-layer k- ε method are discussed, since as far as the authors' survey concerns, the model has been found to involve the least assumption in near-wall modeling and the most suitable for the present numerical test case. Computations were performed using threedimensional code, which is basically the same code as that for ship flow computations. Box-type solution domain is defined; and the inlet and exit boundaries are located at one plate length upstream the leading edge, and two plate lengths downstream the trailing edge, respectively. Number of computational grid is 100×5×100 in longitudinal, transverse, and normal directions, respectively; in which the grid clustering

near the wall boundary was done by the geometry In use of two-layer method, series expansion. minimum grid spacing is crucial, so that the first grid point off the body surface is located in the viscous sublayer, i.e., $y^+ \cong 0.1$. The computational values of Rn are 10⁶, 10⁷, 10⁸, and 10⁹, i.e., the largest Rn is considered to be full-scale in the present study. Solution convergence was critically evaluated, and that was achieved at 500 global iterations. Numerical uncertainties U_{SN} for velocity and resistance are less than 0.5%. The procedure to evaluate U_{SN} followed that demonstrated by the present author¹³⁾. Note that surface roughness effects are not considered in this section.

Fig.4 shows comparison of frictional resistance. The frictional formulas, which are commonly used in ship design, i.e., ATTC (or Schoenherr), ITTC, and Granville lines, are also included. In the Rn range shown in the figure, ATTC line gives the lowest values among the three lines for the same Rn, however, differences among the three gradually decrease as Rn increases. At lower Rn, two-layer $k - \varepsilon$ results $(2Lk \cdot \epsilon)$ are nearly identical to ATTC line; however, the discrepancies tend to increase as Rn increases, e.g., $2Lk \epsilon$ value for $Rn=10^9$ is about 1.4% and 1.3% higher than those of ATTC and ITTC lines, respectively. Note that those frictional formulas are based on measurements for $Rn \sim O(10^8)$ at most, and especially ATTC line has been suspected to give lower value than the real at full-scale Rn. The differences between the present results and the lines at full-scale Rn may be noteworthy, and more detailed investigation on numerical results as well as the frictional formulas has been completed by the present authors14).

Fig.5 provides comparison of near-wall profiles of velocity and turbulence quantities, i.e., y^+ vs. u^+ $(=u'U_r)$, k^+ $(=k'U_r^2)$, ε^+ $(=\varepsilon'RnU_r^4)$, and $v_{\ell'\nu_0}$ all of which are dimensionless parameters. Note that X=0.5 corresponds to midsection of flat plate. In addition, Fig.6 shows y^+ vs. y^+du'/dy^+ , whose value is supposed to approach $1/\kappa$ in the logarithmic region. It is shown in u^+ and y^+du''/dy^+ profiles that the range of y^+ , where the logarithmic law applies, increases as Rn increases. The outer edge of logarithmic region for $Rn=10^9$ is located around $y^+=10^4$. It is also displayed that, in the logarithmic region, k^+ , ε^+ , and $\nu_0^{\prime}\nu_0^{\prime}$ profiles approach the relations $k^+=1/C_{\mu}^{0.5}$, $\varepsilon^+=1/\kappa y^+$, and $\nu_0^{\prime}\nu_0^=\kappa y^+$, respectively, where those are based on the local-equilibrium assumption. For Rn higher than that of the most well-know Nikuradse's experiments, i.e., $Rn=3.4\times10^6$, the validity of logarithmic law had often been argued, until the 'super-pipe' experiments performed at Princeton University clearly indicated the existence of logarithmic-law region for the higher Rn^{20} . The experiments also displayed growing log region as Rn increases, which was also verified in the present results for flat plate flow.

In fact, the trends displayed in the present results are very favorable for use of wall-function method for full-scale Rn, which is based on the following reasons: (1) the extended logarithmic region will reduce miss-arrangement of grid points that often occurs in use of wall-function for model-scale Rn, where the logarithmic region is considerably narrow; (2) large logarithmic region enables to use larger minimum grid spacing, which will reduce numerical unstability that often occurs with small grid spacing for high Rn; and (3) validity of the local-equilibrium assumption shown in profiles of k^+, c^+ , and ν_0/ν_0 is evident for full-scale Rn. Based on the results, discussions on wall-function approach are continued below.



Fig.4 Frictional resistance coefficients for flat plate (2L $k \cdot \varepsilon$ for two-layer $k \cdot \varepsilon$ model; and WF $k \cdot \varepsilon$ for wall-function $k \cdot \varepsilon$ model).



Fig.5 Comparison of near-wall profiles of velocity and turbulence quantities (flat-plate flow, two-layer k- ε model).



Fig.6 Evaluation of logarithmic region (flat-plate flow, two-layer k-s model).

Computations with wall-function k-c method were performed for $Rn=10^7$, 10^8 , and 10^9 . The computational grid was generated in similar manner to that for $2Lk \epsilon$ case, except for minimum grid spacing which is much larger for this case, i.e., that is around $y^{+}=100$, since the near-wall boundary grids can be located in logarithmic region. Solution convergence was achieved as fast as the case for 2Lk- ε , i.e., 500 global iterations were sufficient for all three Rn, and estimated U_{SN} for velocity and resistance are less than 0.5%. Figs.4 and 7 show comparison of resistance and near-wall profiles of dimensionless velocity and turbulence parameters, respectively, where it is shown that the present wall-function method successfully reproduced outer-layer flow of the 2Lk-& results, i.e., boundary layer thickness and range of logarithmic region are

nearly identical; in which the slight differences of the solutions observed near the edge of boundary layer are likely attributed to differences of grid density in the region. An important fact is that almost 50% of total grid points required for $2Lk \cdot \varepsilon$ method can be excluded if the present wall-function is applied, since those grids are located in near-wall layer (see Fig.2).

The reduction of grids yields considerably high computational efficiency, and is more suitable in practical design. All of the above-mentioned are further encouraging to employ wall-function method for full-scale ship-flow simulation, which is demonstrated as described below.



Fig.7 Comparison of near-wall profiles of velocity and turbulence quantities (flat-plate flow, wall-function $k \cdot \epsilon$ model).

4. Ship Flow

In this section, results for full-scale ship flow are discussed. Series 60 C_B =0.6 ship model was selected for the present test case, because the model has been used for many numerical studies and an extensive set of experimental data are available (see reference ¹⁰⁾ for more details of the data). Since the experimental data were obtained for model scale, i.e., average Rn is 2×10^6 , two scaling methods are used to estimate full-scale flow from the data, i.e., Tanaka and Himeno Corrections¹⁵⁾. Tanaka proposed a scaling formulas $\delta \propto [C_F^{1/2} \sim C_F^{1/3}]$ for the wake thickness,

and $(U_e - U)/U_e \propto [C_F^{1/2} \sim C_F^{1/3}]$ for the velocity defect profile, where U_e is edge streamwise velocity, in which the correlations for asymptotic twodimensional and axisymmetric wakes are considered. On the other hand, Himeno suggested $\delta \propto C_F^{0.7}$ and $(U_e \cdot U)/U_e \propto C_F^{1/2}$, which are based on integral correlations developed three-dimensional for turbulent boundary layers. In the present study, scaling relations $(U_e \cdot U)/U_e \propto C_F^{1/2}$ and $(U_e \cdot U)/U_e \propto$ $C_{F^{1/3}}$ are used to identify lower and higher bounds of the correction methods, and hereafter those are referred to as "Himeno Correction" and "Tanaka Correction," respectively.

-325-



Fig.8 Reynolds number scaling of axial velocity (Series 60 ship model; Himeno Correction: $(U_e \cdot u)/U_e \propto C_F^{1/2}$).



Fig.9 Reynolds number scaling of axial velocity (Series 60 ship model; Tanaka correction: $(U_e \cdot u)/U_e \propto C_F^{1/3}$).



Fig.10 Comparison of axial-velocity contours between computations and experiments for model-scale Reynolds number (Series 60 ship model, $Rn=2\times10^6$).

-326-

96



Fig.11 Comparison of axial-velocity contours between computations and experiments for full-scale Reynolds number (Series 60 ship model, $Rn=2\times10^9$).

Figs.8 and 9 show comparison of axial-velocity (U) contours for the sections near the stern, i.e., X=0.9 (S.S.1) and X=1.0 (AP), for those correction methods, respectively. It is important to note that, although these scaling laws are widely accepted by practical design, the predicted shrinkages of boundary layer flow associated with increase of Rn are mostly too uniform. The larger shrinkage is predicted by the Himeno Correction, that is considered to be more appropriate in thinner turbulent boundary layer.

Computation was made for full-scale Rn, i.e., Rn= 2×10^9 , by using two-point wall-function method, which is based on the results described in the previous section. Cylindrical computational domain is used, where the inlet and exit boundaries are located at one ship length upstream the FP (X=0), and two ship lengths downstream the AP (X=1), respectively. Computational grid is 240×30×40 in longitudinal, girthwise, and radial directions, respectively; and the first and second grid points off the body surface are located in the logarithmic layer, i.e., $y^+ \cong 10^3$. As is the case for flat plate, the grid clustering near the wall boundary was done by the geometry series expansion. This grid arrangement enabled to exclude many grids, which were considered to be located in near-wall region in use of two-layer or any other similar modeling.

As is the case for flat plate, convergence of solutions was critically evaluated, and that was achieved at 1000 global iterations. A representative

 U_{SN} for velocity field is that of the center of propeller disk, which is found to be about 1%. Also, U_{SN} for resistance coefficients appeared to be about 1.5%. Note that surface roughness effects are still not considered. In the following discussions, the model-scale results for the same ship model by the present author ($Rn= 2 \times 10^6$)¹¹ are also included for The results were obtained from comparison. two-layer $k - \varepsilon$ method, which appeared superior to wall-function method for the scale (the reason was also implicated by the present results for flat plate, i.e., very narrow logarithmic region for the scale), and overall agreement with measurements was shown satisfactory for the possibly maximum capability of the turbulence model in application to ship flow. More details were discussed by the author¹¹⁾.

Figs.10 and 11 show comparison of model-scale and full-scale axial-velocity (U) contours, respectively, for the sections near the stern, i.e., X=0.9 and X=1.0. Some flow aspects for full-scale results are similar to those for model scale, i.e., for X=0.9, the U contours display a pronounced bulge in the boundary layer near the region of maximum hull concavity and reduction in the boundary layer near the centerplane; and for X=1.0, the results display merging of the boundary layer into the wake and its initial evolution. The differences between full- and model-scale results are mainly attributed to evident shrinkage in viscous region. For full-scale results at X=0.9, the computed contour lines are mostly closer to Himeno

-327-

Corrections; however, at X=1.0, i.e., in the near wake, those are closer to Tanaka Correction. The trends shown in the U contours are consistent with the theory used in the two correction methods.



Fig.12 Comparison of surface pressure (Cp) contours between for model- and full-scale Reynolds numbers (Series 60 ship model).





Fig.13 Comparison of shear stress contours $(C\tau_X)$ between for model- and full-scale Reynolds numbers (Series 60 ship model).

A noteworthy feature displayed in the results is that the shrinkages of boundary layer flow are generally not as uniform as predictions by the scaling laws. Although the complete justification of computed results may be difficult, this likely indicates some limitations of simple scaling methods in detailed prediction of Rn influences on flows in near-wake region. The same implication was also confirmed by others³⁾.

Figs. 12 and 13 show comparison of surfacepressure (Cp) and shear-stress $(C\tau_X)$ distributions

between for the model and full-scale results. $C_{\tau x}$ values for full-scale results are clearly lower on all over the surface. On the other hand, Cp contours are mostly similar on the forebody between for the two results; however, the differences turn out to be clearer in midgirth region near the stern and near-wake region. As other study for the tanker hull form³⁾ suggested, the differences in Cp are seen to be too evident to totally ignore scale effects on form factor. The form factor K predicted by the present full-scale simulation is about 0.20 based on ATTC (or Schoenherr) value. The value of K appears slightly larger than that of the model-scale measurements, i.e., 0.18; and the trend further supports the underestimation possibly occurs in ATTC values for full scale. Currently, computations for tanker hull form are in progress using the present method, that, possibly in combination with reports from others, will enable more detailed investigation on Rn influences on form factor.

5. Consideration of Surface Roughness Effects

Finally, the inclusion of surface roughness effects on flow and resistance is discussed in this section. Computations were made for flat plate using wall-function $k - \varepsilon$ method. The full-scale condition is assumed, i.e., the computational Rn is 10⁹. The uniform sand grain roughness was considered, and the value was taken to be $k_s = 0 \sim 5 \times 10^{-7}$. Figs.14 through 16 show comparison of near-wall profiles of dimensionless velocity and turbulence parameters, frictional drag coefficients, and frictional velocity distributions, respectively. An advantage of the present wall-function method is that surface roughness effects are easily considered by giving k_{\star} to equation (8), and no other numerical model is required.

As shown in Fig.14, logarithmic line goes downward as k_s increases, and profiles of turbulence parameters, i.e., k^+ , ε^+ , and ν_0/ν_0 , shift rightward, which is mainly due to the increase of frictional velocity. For all cases shown in the figure, k^+ , ε^+ , and ν_0/ν_0 profiles in the logarithmic region approach the relations $k^+=1/C_{\mu}^{0.5}$, $\varepsilon^+=1/\kappa y^+$, and $\nu_0/\nu_0=\kappa y^+$, respectively; all of which indicate that the

- 328 -





Fig.14 Surface roughness effects on near-wall profiles of velocity and turbulence quantities (flat-plate flow, wall-function $k \cdot \varepsilon$ model, $Rn=10^9$).



Fig.15 Surface roughness effects on frictional resistance (C_F) (flat-plate flow, wall-function k- ε model, Rn=10⁹).



Fig.16 Surface roughness effects on frictional velocity (U_r) (flat-plate flow, wall-function k-c model, $Rn=10^9$).

-329-

trends shown in the present results agree well with theory widely accepted for the roughness level, i.e., that based on the validity of the local-equilibrium assumption in the logarithmic region for the case with rough wall.

On the other hand, as shown in Fig.15, frictional resistance rapidly increases as k_s increases, that is consistent with physical intuition. For instance, if the 10% increase in frictional resistance due to the roughness is assumed, the estimated k_s from the present results is around 2×10⁻⁸ (Fig.15), for which Fig.16 indicates that k_{*}^{+} is clearly less than 5, i.e., the roughness is considered to be still hydrodynamically smooth level. Noted that ITTC (1978) recommended different definition of k, and method to give ΔC_F , i.e., $\Delta C_F = (105k_S^{1/3} - 0.64) \times 10^{-3}$, and *Rn* influences on ΔC_F are ignored. The ITTC method predicts constant $\Delta C_F = 1.53 \times 10^{-4}$, i.e., frictional resistance increases in 10% for $Rn=10^9$ as assumed in the above discussion, if the k_s is taken to be 4.3×10⁻⁷, which is larger value as compared to the present results. Although the differences in definition of k_s must be reconsidered, further evaluation of the present method is of great interest in comparison to the method of ITTC, and also of interest is evaluation of surface roughness effects on propulsion performance for arbitrarily specified k_s on hull surface.

6. Concluding Remarks

The present study concerns simulation of ship viscous flow at full-scale Rn, where main focuses have been placed on development of RaNS equation method applicable to full-scale flow simulation and investigation on appropriate physical model for full-scale Rn in conjunction with consideration of near-wall flow model including surface roughness effects. The present numerical method is based on extension of method developed by the present author, such that, in association with standard k model, two near-wall models can be employed, i.e., two-layer method and two-point wall-function method.

It has been shown that the present method successfully yields solutions in the range from modelto full scale Rn, and for the first time, solutions of two-layer k model have been presented for the range of $Rn=10^6 \sim 10^9$. Detailed investigation on near-wall flow for flat plate revealed that logarithmic region clearly increases as Rn increases, and the outer edge of logarithmic region for Rn=10⁹ is located over $y^{+}=10^{4}$. The trends displayed in the two-layer $k \varepsilon$ results are very favorable for use of wall-function method for full-scale Rn, which has been verified in application to ship flow as well as flat-plate flow. Especially for full-scale, the present two-point wall-function method appeared more suitable in practical design use for the higher computational efficiency and capability to include surface roughness effects. Furthermore, the present method has shown capable for detailed studies on Rn influences on flow and resistance including that of surface roughness effects, where all are of great interest in comparison to simple scaling laws or conventional empirical formulas.

Further validation of the present method is currently in progress, where application to tanker hull form and more detailed investigation on roughness effects on ship propulsion performance are involved, all of which will be reported in future publications.

Acknowledgement

The present work has been partly supported by Grant-in-Aid for Scientific Research (2000-2001, Project Number 12650903). The authors wish to thank whom concern the program for their kind support and encouragement.

References

1) Webster, J., and Mutnick, I.: Future Surface Combatant, Proceedings 25th ATTC Conference, Iowa City, IA, September, 1998.

2) Patel, V.C.: Flow at High Reynolds Number and over Rough Surfaces - Achilles Heal of CFD, Lecture Note, 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, May 25-27, 1998.

3) Ju, S. and Patel, V.C.: Stern Flows at Full-Scale Reynolds Number, J. Ship Research, Vol. 35, No. 2,

-330-

1991, pp.101-113.

4) Shirose, Y. and Masuko, A.: Numerical Calculation of Viscous Flow around a Full Scale VLCC and Its Geosim Models, Proc. 2nd Osaka Colloquium on Viscous Fluid Dynamics in Ship and Ocean Technology, Osaka, Japan, 1991, pp. 271-290.

5) Ishikawa, S.: Study on Scale Effect on Viscous Flow around Hull and on Propulsive Performance of a Ship by Using CFD, J. West Japan Society of Naval Architects, No. 91, 1995, pp.1-14.

6) Shingo, S. and Ikehata, M.: CFD Simulation of Flow Field around Ship Hull at High Reynolds Number, J. Kansai Society of Naval Architects, Japan, No. 233, 2000, pp.11-16.

7) Proc. Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics, Gothenburg, 2000.

8) ITTC Report of Resistance and Flow Committee, Proc. 20th International Towing Tank Conference, 1993, pp. 17-66.

9) Coder, D.W.: Mean Velocities in the Turbulent Boundary Layer Flow of an Appended Body of Revolution for Reynolds Numbers from 20 Million to 1000 Million, Proc. 17th Office of Naval Research Symposium on Naval Hydrodynamics, The Hague, The Netherlands, 1988, pp. 289-297.

10) Toda, Y., Stern, F., and Longo, J.: Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B =.6 Ship Model \cdot Part 1: Froude Numbers .16 and .316 \cdot , J. Ship Research, Vol. 37, No. 4, 1992, pp.360-377.

11) Tahara, Y.: An Application of Two-Layer $k \cdot \varepsilon$ Model to Ship Flow Computation, J. Society of Naval Architects, Japan, Vol. 177, 1995, pp. 161-176.

12) White F.M.: Viscous Fluid Flow, McGraw Hill, New York, NY, 1991.

13) Tahara, Y.: Wave Influences on Viscous Flow around a Ship in Steady Yaw Motion, J. The Society of Naval Architects of Japan, Vol. 186, 1999, pp. 157-168.

14) Katsui, T., Himeno, Y., and Tahara, Y.: On Reconsideration on Flat Plate Frictional Resistance, to be presented at the Joint Autumn Meeting of Three Societies of Naval Architects in Japan, November, 2002.

15) Tanaka, I.: Three-Dimensional Ship Boundary Layer and Wake, Advances in Applied Mechanics, Vol.26, 1988, pp.311-359.

-331-

平板摩擦抵抗値の再検討について*1

正会員勝井 辰博 •²,正会員姫野 洋司 •² 正会員田原 裕介 •²

On Reconsideration on Flat Plate Frictional Resistance

By Tokihiro KATSUI (Member), Yoji HIMENO (Member)

and

Yusuke TAHARA (Member)

This paper presents a new friction coefficient of flat plate in wide Raynols number range. The main purpose of this study is to present a verification data for CFD calculation, which will be applied to predict ship performance in full scale Reynols number. The friction coefficient is calculated by solving a differential equation without approximation that is based on momentum equation and Coles' wall-wake law. The parameters used in Coles' wall-wake law for the assumption of the velocity profile in turbulent boundary layer are determined based on reliable experimental data by Osaka et al. The calculated results of flat plate frictional resistance, local frictional resistance and velocity profile in turbulent boundary layer agree well with experimental ones.

However, few experimental data aviod to verify the results at high Reynols number. therfore the research should be continued focusing on the behavior of wake function at high Reynols number.

Keywords : Friction Coefficient, Flat Plate. Boundary Layer Theory, Schoenherr's Formula, Verification of CFD

1. 緒言

本研究は平板の摩擦抵抗係数について境界層理論に 基づいた再検討を行い、幅広いレイノルズ数の範囲に おける摩擦抵抗係数の推定値を新たに示したものであ る、本研究の目的は、外挿法による船舶の馬力推定の ために必要な相当平板の摩擦抵抗係数値を正確に算 定することだけではなく、近年盛んに行われるように なった CFD による船舶の性能推定計算に対して検証 データを提供することにある、今後、実船スケールの レイノルズ数での CFD 計算が多く行われることが予 想されるから、特に実験結果の乏しい高レイノルズ数 での摩擦抵抗値について信頼性の高い検証データを示 すことが重要である、

日本の造船所においては古くから摩擦抵抗の算定に Schoenherr¹⁾の式が利用されてきたため、CFD計算 結果の検証においてもこの式が用いられることが多 い. しかし Schoenherr の式はあくまで実験公式であっ てばらつきのあるデータの平均的な値を取っている こと、また実船スケールのレイノルズ数における実 験データがないことに問題がある.ヨーロッパにおい ては ITTC が 1957 年に提案したいわゆる correlation line が標準的に用いられているようであるが、これに 対し近年 Grigson²⁾ は境界層理論と種々の実験データ にもとづいて算定された新しい摩擦抵抗係数値を提案 している. 運動量積分式に基づくこの方法では、摩擦 抵抗係数の精度は境界層内の流速分布の推定精度に依 存するため、速度分布に関する仮定の検証は重要であ る.大坂ら3)は平板摩擦抵抗の計測に際し、流れの2 次元性の確保について厳密に検討を行っている. 本研 究では、大坂らの実験による平板の乱流境界層内速度 分布に関する知見を援用して運動量積分方程式に基づ く厳密な微分方程式を解き、広いレイノルズ数の範囲 で平板の摩擦抵抗係数を算定した.

^{*1} 平成 14 年 11 月 14 日 造船三学会連合大会において講演, 原稿受付 平成 14 年 11 月 29 日

^{*2} 大阪府立大学大学院工学研究科

2. 理論

2.1 運動量積分式

運動量積分式に基づいた平板摩擦抵抗係数の算定を 行う.なお、本論に用いる記号の定義は文末に示すと おりである.2次元の圧力勾配のない平板周りの流れを 考えると、運動量積分式は以下のようになる

$$\frac{d\theta}{dx} = \frac{1}{2}C_f \tag{1}$$

これを x = 0 すなわち平板前緑で運動 量厚さ θ が 0 と なる条件のもとに積分すれば

$$\theta = \frac{1}{2}C_F x \tag{2}$$

となる.この運動最厚さの境界層厚さに対する比は、摩 擦速度を用いた無次元最によって以下のように表される.

$$\frac{\theta}{\delta} = \frac{1}{2} C_F \frac{Rn \cdot \sigma}{\delta^+} \tag{3}$$

一方、運動量厚さの定義式は

$$\theta \equiv \int_0^\delta \frac{u}{U} \left(1 - \frac{u}{U} \right) dy \tag{4}$$

であり、序擦速度を用いた無次元量によって

$$\frac{\theta}{\delta} = \frac{1}{\delta^+} \int_0^{\delta^+} \sigma u^+ \left(1 - \sigma u^+\right) dy^+ \tag{5}$$

と表される. これにより (3), (5) 式から

$$\int_{0}^{\delta^{+}} u^{+} \left(1 - \sigma u^{+}\right) dy^{+} = \frac{1}{2} C_{F} R n \qquad (6)$$

の関係が得られる.ここで, 無次元摩擦速度 σ と局所 摩擦抵抗係数 C_f は

$$\sigma \equiv \frac{u_{\tau}}{U} = \sqrt{\frac{C_f}{2}} \tag{7}$$

の関係にある. また (1), (2) 式から平板摩擦抵抗係数 と局所摩擦抵抗係数は

$$C_{f} = C_{F} + \frac{dC_{F}}{dx}$$
$$= C_{F} + Rn\frac{dC_{F}}{dRn}$$
(8)

と関連付けられるので、無次元摩擦速度は

$$\sigma = \sqrt{\frac{C_F + Rn \frac{dC_F}{dRn}}{2}} \tag{9}$$

と表される.(6),(9) 式は乱流境界層内の流速分布が分 かれば、平板摩擦抵抗係数 C_F に関する微分方程式を 与えるものである。

2.2 乱流境界層内速度分布

平板乱流境界層内の流速分布は Fig. 1に示すような 摩擦速度を用いた相似則が知られている。図中に示し たように I. 粘性応力が卓越する直線低層, II. 粘性応力 とレイノルズ応力が同程度のパッファー層, III. レイノ ルズ応力が卓越する対数領域から外層。の3領域に分 割すれば、それぞれの領域での流速分布は以下のよう に与えられる。



Fig. 1 Time averaged structure of turbulent boundary layer.

$$I. u^+ = y^+ (10)$$

II.
$$\begin{cases} \frac{du}{dy^+} = \frac{1}{1 + \nu_t/\nu} \\ \frac{\nu_t}{\nu} = \kappa \left(y^+ - \lambda_1 \tanh(y^+/\lambda_1) \right), \ \lambda_1 = 11 \end{cases}$$
(11)

$$\Pi. \begin{cases} u^{+} = \frac{1}{\kappa} \ln(y^{+}) + C + \frac{\Pi}{\kappa} w(y^{+}/\delta^{+}) \\ w(y^{+}/\delta^{+}) = 1 - \cos(\pi y^{+}/\delta^{+}) \end{cases} (12)$$

(11) 式は Reichardt⁴⁾の式,(12) 式は Coles の式であ る.式中の κ はカルマン定数,C は対数則の切片,П は後流パラメタと呼ばれ実験等により定められる.(12) 式が適用される範囲は境界層の最下層を除く大部分で あるため、摩擦抵抗値はこれらの値に大きく依存する. したがってこれらのモデル係数の値の設定には注意を 要する.大坂ら³⁾は圧力勾配のない滑面平板乱流境 界層において運動量積分式を用いて流場の2次元性 が良好であることを確認し、 $Rn_{\theta} = 840 \sim 6220$ の範囲 ($Rn = 2.8 \times 10^5 \sim 3.5 \times 10^6$ 程度と推定される)で直接 測定装置によって局所摩擦抵抗係数を求めた.彼らは 既存の実験データとの比較を行い、流場の2次元性の 確保の程度が摩擦抵抗の計測値に影響することを指摘 している.この中で速度分布について以下のような結 論を示している.

i) 計測したすべてのレイノルズ数の範囲において対

数直線部が存在し、その傾きであるカルマン定数 κの値は 0.41 である.

- ii) 対数則の切片 C は低レイノルズ数でレイノルズ
 数が減少するにしたがって増加する.
- iii) 後流パラメタ Π は (13) 式で表される Coles の式で よく近似でき高レイノルズ数で 0.62 に漸近する.

$$\Pi = 0.62 - 1.21 \exp\left(-\delta^{+}/290\right) \tag{13}$$

大坂らの実験のレイノルズ数は模型船の範囲の域を越 えないものであるが、本研究ではこれらの結果を参考 にして、 $\kappa = 0.41$, C = 5.0を採用し、後流パラメタ Пにつては (13) 式を用いることとした. このように κ , C, Π を与えれば、(10)~(12) 式を積分し、運動量 厚さを求めることができる. (10)~(12) 式の適用範囲 については以下のように定めた. $0 \le y^+ \le 5$ では (10) 式を用い、それ以後 $y^+ = 5$, $u^+ = 5$ から (11) 式に基 づいて積分して u^+ を定め、この値が (12) 式の値と一 致したところ ($y^+ = 27.2$) から以降は (12) 式を用いる.

2.3 微分方程式

ここで $F_1(\delta^+)$, $F_2(\delta^+)$ を以下のように定義する.

$$F_{1}(\delta^{+}) \equiv \int_{0}^{\delta^{+}} u^{+} dy^{+}$$

$$= \int_{0}^{y_{1}^{+}} y^{+} dy^{+}$$

$$+ \int_{y_{1}^{+}}^{y_{2}^{+}} \left(\int_{y_{1}^{+}}^{y^{+}} \frac{dy^{+}}{1 + \kappa (y^{+} - \lambda_{1} \tanh(y^{+}/\lambda_{1}))} + y_{1}^{+} \right) dy^{+}$$

$$+ \int_{y_{2}^{+}}^{\delta^{+}} \left(\frac{1}{\kappa} \ln(y^{+}) + C + \frac{\Pi}{\kappa} \left(1 - \cos\left(\pi \frac{y^{+}}{\delta^{+}}\right) \right) \right) dy^{+}$$

$$\cdots (14)$$

$$F_{2}(\delta^{+}) \equiv \int_{0}^{\delta^{+}} (u^{+})^{2} dy^{+}$$

= $\int_{0}^{y_{1}^{+}} (y^{+})^{2} dy^{+}$
+ $\int_{y_{1}^{+}}^{y_{2}^{+}} \left(\int_{y_{1}^{+}}^{y^{+}} \frac{dy^{+}}{1 + \kappa (y^{+} - \lambda_{1} \tanh(y^{+}/\lambda_{1}))} + y_{1}^{+} \right)^{2} dy^{+}$
+ $\int_{y_{2}^{+}}^{\delta^{+}} \left(\frac{1}{\kappa} \ln(y^{+}) + C + \frac{\Pi}{\kappa} \left(1 - \cos\left(\pi \frac{y^{+}}{\delta^{+}}\right) \right) \right)^{2} dy^{+}$
... (15)

ただし、
$$\begin{cases} y_1^+ = 5.0, \ y_2^+ = 27.2, \ \lambda_1 = 11, \\ \kappa = 0.41, \ C = 5.0 \\ \Pi = 0.62 - 1.21 \exp(-\delta^+/290) \end{cases}$$

数直線部が存在し、その傾きであるカルマン定数 である。このとき、(6)式は以下のように表される.

$$F_{1}(\delta^{+}) - \sqrt{\frac{C_{F} + Rn\frac{dC_{F}}{dRn}}{2}}F_{2}(\delta^{+}) = \frac{1}{2}C_{F} Rn \quad (16)$$

(16) 式のみではδ⁺ を定めることができない. (12) 式 の流速分布において境界層外端で、流速が一様流速と なることを考慮すれば

$$\frac{1}{\sigma} = \frac{1}{\kappa} \ln(\delta^+) + C + \frac{\Pi}{\kappa} w(1)$$
 (17)

となる. 具体的には

$$\sqrt{\frac{2}{C_F + Rn \frac{dC_F}{dRn}}} = \frac{1}{\kappa} \ln(\delta^+) + C + \frac{2}{\kappa} \left(0.62 - 1.21 \exp(\delta^+/290)\right) (18)$$

を得る. (16), (18) 式は Rn および C_F を与えたとき, dC_F/dRn および δ^+ を未知とする連立方程式, すなわ ち微分方程式である. 適当な初期値, Rn および C_F を 与えれば dC_F/dRn および δ^+ が得られ, ルンゲクッタ 法等により各 Rn に対する C_F の値を算定できる. た だし, (8) 式にもとづいて П を定める場合, レイノル ズ数が小さく δ^+ が小さな値となる場合, П の値が負 となって不都合である (具体的には δ^+ < 193 の場合). 実験結果によれば低レイノルズ数において П ははほぼ 0 である. ここでは得られた δ^+ が 193 以下の場合は П = 0 とした.

3. 計算結果と考察

3.1 微分方程式の初期値依存性

微分方程式を解くにあたり初期値を設定する必要が あるため、微分方程式の初期値依存性についての調 査を行った.具体的には初期値として Rn = 1.0 × 10⁴ に対して Schoenherr の式より求められた平板の全摩 擾抵抗係数の値、およびその 1.1 倍と 0.9 倍の値の 3 種類の初期値を与えたときの計算を行った。この結果 をFig. 2に示す. それぞれの初期値に対する解はレ イノルズ数が増加するにしたがって同じ値に漸近し, $Rn = 1.0 \times 10^5$ まで達するとほぼ差がなくなることが 分かる。一方、この3種類の初期値に対する局所摩擦抵 抗係数の計算結果は Fig. 3のようになる. 平板の全摩 擦抵抗係数 Cr の初期値を大きく設定すると局所際擦 抵抗係数 C_f は小さな値をとり、逆に C_F の初期値を 小さく設定すると Cf は大きな値をとる. この差もレ イノルズ数が大きくなるにしたがって小さくなり、CF の計算結果の差がなくなるあたりでCfの差もなくな る. すなわち, CF の計算結果に対する初期値影響は



Fig. 2 Dependency of initial value to the solution of differential equation.



Fig. 3 Dependency of initial value to the calculated results of local friction coefficient.

 C_f の差によって相殺され、レイノルズ数が大きくなる とその影響がなくなるといえる.このため本論では $Rn = 1.0 \times 10^4$ で Schoenherr の式より求められた C_F を初期値として計算した結果について $Rn = 1.0 \times 10^5$ 以上のものを採用することにする.

局所摩擦抵抗係数 C_f の計算結果には $Rn = 1.0 \times 10^5$ 付近で変化が滑らかではない点が存在するが、これは 後流パラメタ Π の値として δ^+ が 193 以下の場合は 0、 それ以上の場合は (13) 式に基づいた値を採用している ためであり、この点はちょうど $\delta^+ = 193$ になるレイノ ルズ数である。後流パラメタの値について 0 から滑ら かに (13) 式に接続するように取り扱えばこのようなこ とは起きないと考えられるが、平板摩擦抵抗係数その ものは局所摩擦抵抗の積分値であるため滑らかな曲線 であり、大きな影響はないと考える。

3.2 境界層内流速分布

(16), (18) 式より δ⁺ が確定すれば.境界層外端位置 と後流バラメタ Π が確定するので.(10)~(12) 式によ り境界層内流速分布が得られる。この結果を大坂らの 実験結果と比較したものを Fig. 4に示す。大坂らの実 験では運動最厚さに基づくレイノルズ数 Rn_{θ} を用いて いるため、計算結果もこれに合わせた。図中に示し た $Rn_{\theta} = 840, 1230, 2100, 2990. 4400, 5230. 6040$ に対応するレイノルズ数は計算結果において は $Rn = 2.77 \times 10^5, 4.58 \times 10^5, 9.22 \times 10^5, 1.45 \times 10^6,$ $2.36 \times 10^6, 2.93 \times 10^6, 3.50 \times 10^6$ である。各レイノル ズとも粘性低層から境界層外端に至るまで両者はよく 一致している。



Fig. 4 Comparison of velocity profile. (Experimental data are obtained by Osaka et al.³⁾)

3.3 局所摩擦抵抗係数

Fig. 5に局所摩擦抵抗係数について計算結果と大坂 らの実験結果との比較を示す。図中の上段にレイノル ズ数に対応した値を、下段に運動量厚さに基づくレイ ノルズ数に対応した値を示してある。白丸が大坂らの 直接計測の値であり、二点鎖線が本論の計算結果であ る。大坂らの実験値は図中にもあるように Schoenherr の式および他者の実験結果と比較し小さい。この理由 を大坂らは、他者の実験では流場の 2 次元性が確保さ れていないためであるとしている。本論の結果はレイ ノルズ数が小さいところを除いて大坂らの値とほぼ一 致している。

Fig. 6に高レイノルズ数における局所摩擦抵抗係数 の計算結果を Grigson²⁾の計算結果に重ねたものを示 す. 本研究の値は総じて Grigson のものより小さく、 Kempf の実験結果とよい一致を示す. Grigson の値は むしろ supersonic での実験結果に近い.



Fig. 5 Comparison of local friction coefficient. (Experimental data are obtained by Osaka et al.³⁾)



Fig. 6 Comparison of local friction coefficient with Grigson's results.



Fig. 7 Comparison of friction coefficient of flat plate.

3.4 平板摩擦抵抗係数

Fig. 7に平板の全摩擦抵抗係数値の計算結果を示 す. 本論および Grigson²⁾の値の Schoenherr の値に 対する比をあわせて図示してある.本論の結果は $Rn = 1.6 \times 10^6$ を超えるあたりから Schoenherr の値 よりも小さな値をとり Rn = 7.0 × 10⁶ 付近でその比 が最小になる. このとき本論の結果は Schoenherr に 比べて約1.6%小さい. その後,緩やかに Schoenherr の値に近づき、Rn = 8.0×10⁶ 付近で一致し、それ 以降 Schoenherr の値よりも大きくなる. その差は 徐々に大きくなり Rn = 1.0 × 10¹⁰ では約 2.4%とな る. 一方 Grigson の値は Rn = 1.5~5.0×10⁶ 付近で 本論の結果よりもさらに小さい. Schoenherr の値と 比較すると Rn = 3.0 × 10⁶ 付近においてその差は最 大となり、Schoenherr の値よりも約3%程度小さくな る. その後 Schoenherr の値に近づき, $Rn = 9.0 \times 10^6$ 付近で一致した後、Schoenherr の値より大きくなり $Rn = 2.0 \times 10^7$ では約 2%大きくなる. $Rn = 1.0 \times 10^8$ 以上では常に本論の結果よりも大きく, Rn = 4.0 × 109 では Schoenherr に比して 6%大きな値となる。高レイ ノルズ数での摩擦抵抗の計測を行った Kempf⁵⁾の実験 との比較を Fig. 8に示す. Schoenherr の値と本論の結 果にそれほど大きな差はないが、本論の結果が Kempf の実験結果に最も近い値をとっていると思われる、本

論と Grigson の計算結果の違いは速度分布則に用いる モデル定数の違いや計算手法の違いによるものと考え られるが、今後詳細な比較検討が必要であると考える、

また、本論では Π の値として (13) 式を採用してい る.これは $Rn = 4.0 \times 10^6$ 程度までの実験結果では検 証されているものの、高レイノルズ数については不明 である、高レイノルズ数において本論で用いた値は (13) 式の漸近値である 0.62 であり、これはあくまで外 挿に過ぎないが、現在のところ高レイノルズ数の極限 で Π の値が漸近しない状況は想定し難い、また $\kappa \approx C$ の値はあくまで壁法則 (低レイノルズ数) で定められる べきであり、高レイノルズ数領域 (10⁶ 以上) での議論 はさほど重要ではないと考える、もっとも、高レイノ ルズ数における Π の漸近値等、境界層外層の挙動につ いては今後 CFD を援用した検討が必要であろう、以 上により、今回の結果は現時点での平板乱流境界層の 研究成果をできる限り反映して実船レベルの高レイノ ルズ数領域まで拡張したものと言える、

4. 結言

本研究は平板の摩擦抵抗係数について運動量積分式 に基づく厳密な微分方程式を解き、平板摩擦抵抗係数 を算定した、境界層内速度分布については大坂らの実 験結果をもとにパラメタを定め、粘性低層から境界層



Fig. 8 Comparison of friction coefficient of flat plate at high Reynols number with experimental results.

外端に至るまでの速度分布を考慮に入れた、得られた 計算結果は大坂らの実験結果に対して低レイノルズ数 の範囲内ではあるものの、境界層内速度分布、局所摩 擦抵抗係数ともによく一致した、高レイノルズ数にお ける値についても計算を行い、Kempfの実験結果に対 して矛盾のない結果を得ている、ただし、高レイノル ズ数での計算に用いた境界層内流速分布のパラメタは 低レイノルズ数における値を外挿したものであり、今 後 CFD 計算等の結果を扱用し本論の計算手法の検証 および改良を行う予定である。

記号表

x,y:主流方向および鉛直方向の座標 U:一様流速 u:主流方向の流速 ρ:流体の密度 ν:流体の動粘性係数 τω:壁面摩擦応力 u₇:摩擦速度 δ:境界層厚さ θ : 運動量厚さ (= $\int_0^{\delta} u/U(1-u/U)dy$) C1:局所摩擦抵抗係数 CF:摩擦抵抗係数 $Rn: 平板長さに基づくレイノルズ数 (= UL/\nu)$ Rn_{θ} :運動最厚さに基づくレイノルズ数 (= $U\theta/\nu$) u⁺: 摩擦速度で無次元化した主流方向流速 (= u/u₊) y⁺:摩擦速度と壁面からの距離に基づくレイノルズ数 $(= u_{\tau}y/\nu)$ δ⁺:摩擦速度と境界層厚さに基づくレイノルズ数 $(= u_\tau \delta / \nu)$ σ : 無次元摩擦速度 (= u_τ/U)

参 考 文 献

- Karl E. Schoenherr : Resistance of Flat Surfaces, Trans SNAME, Vol. 40, pp.279-313, 1932.
- C. W. B. Grigson : An Accurate Smooth Friction Line for Use in Performance Prediction, Trans RINA, PART A, Vol. 135, pp.149-162, 1993.
- 3) 大坂英雄, 色田孝嗣, 望月信介: 滑面乱流境界層の局所壁面摩擦抵抗係数と平均量特性, 日本機械 学会論文集, (B編)62巻598号, pp.138-145, 1996.
- 4) 谷一郎:流体力学の進歩(乱流).丸善.

討

5) G. Kempf : Neue Ergebnisse der Widerstandsforschung, Werft Reederei u. Hafen. June. 1929.

論

【討論】 田村 欣也

新しい平板の際擦抵抗係数を算定された事に敬意を表 します.第8回 ITTC において ITTC 1957 Correlation Line が暫定的に採択されてから既に半世紀近くが経過 しており、これを見直すための新しい試みが是非とも必 要とされる段階に来ております.現在 ITTC において は、Resistance 及び Propulsion Committee において た々Grigson の式が取り上げられておりますが,必ずし も式自体やそれを算出した根拠についての十分な検討 を行う事なしに、その採否の論議が進められる傾向が 見られるようで甚だ憂慮にたえませんでした.この問 題に対する日本の initiative を確保する上からも、今回 の研究は非常に貴重なものと思いますので、是非とも 速やかに英文リポートを関係する ITTC の技術委員会 に送付して、議論を喚起して頂きたくお願い致します. [回答]

【四名】

ありがとうございます。ご提案いただいた英文リポー トを関係する ITTC の技術委員会に送付する件につき まして、今後英文の論文を執筆する方向で検討して まいりたいと考えております。また、ご指摘のとおり Grigson の研究に対する評価、検証を十分に行う必要 があると考えます、今回の我々の結果と比較しますと、 Grigson の平板摩擦抵抗係数の値は、模型レベルより も低いレイノルズ数で我々の値よりは低め、実船レベ ルでは逆に少し高めの値のようであります。このよう に、現状では結果の比較にとどまっておりますが、今 後の研究のなかで十分な検証をしていきたいと考えて おります。ただ、私たちの研究の意図は外挿法に用い るための新しい Friction Line を提案するということで はなく、今後増えるであろう CFD による実船性能推 定に対して摩擦抵抗に関する評価基準を提案すること にあります、その点をご理解いただけますと幸いです. [討論] (大阪大学) 田中 一朗



Fig. 9 Comparison of friction coefficient with Hughes and Wieghardt's fomula.

- (1) 考え方はきちんとした境界層理論に従い、これに コンピュータ時代の計算手法を導入してまとめら れた本論文は、大変與味深く、有益と考えます。
- (2) 船舶関係の論文では2次元性を合理的に検討した ものとして Wieghardt の優れた研究があります. これとの比較を行ってもらいたいと思います。

[回答]

ありがとうございます. ご指摘頂いた, Wieghardt の研究との比較について Fig. 9に Hughes^{A-1)} および Wieghardt^{A-2)}の式との比較を示します. 概観すると, 図示したレイノルズ数の範囲 ($10^6 < Rn < 10^{10}$)で, 本研究の結果は Hughes に比べて C_F の値で 2.0 × 10^{-4} 程度大きく, Wieghardt と比較すると 1.0×10^{-4} 程度 大きくなっております.

- A-1) G. Hughes : Frictional Resistance of Smooth Plane Surfaces in Turbulent Flow - New Data and a Survey of Existing Data, Trans I.N.A. 1952.
- A-2) K. Wieghardt : Uber den Reibungswiderstand von Platten - Bemerkungen zu zwei Arbeiten von G. Hughes. - , Schiff und Hafen 1955, Heft 2.

関西造船協会論文集 第240号 平成15年9月

Development of CAD/CFD/Optimizer-Integrated Hull-Form Design System^{*1}

By Yusuke TAHARA (*Member*)^{*2}, So SUGIMOTO (*Member*)^{*3} Shinya MURAYAMA (*Member*)^{*4}, Tokihiro KATSUI (*Member*)^{*2} and Yoji HIMENO (*Member*)^{*2}

This paper concerns development of a CAD/CFD/optimizer-integrated hull-form design System. The CAD, CFD, and optimizer modules are functionally independent and basically replaceable. Main objective of the present study is system development and demonstration of the capability, which justifies use of relatively simple flow and free-surface models, static sinkage and trim, and simplified design constraints. The CAD and CFD methods used in the present study are NAPA and a Rankine-source panel method, respectively. Currently, two nonlinear optimization algorithms are implemented, i.e., the successive quadratic programming (SQP) and the genetic algorithm (GA); and results are presented for the former in the present paper. For demonstration of the present approach, the bulbous bow optimization of container ship for minimum wavemaking resistance is considered. It appeared that the present CAD-based hull form modification overcomes many issues related to unrealistic solutions often seen in the related studies, which is due to the use of insufficient hull form modification methods. In conclusion, the present system has been shown very promising and warrants further investigation for more practical design constraints and conditions.

Keywords: CAD, CFD, Hull-Form Optimization, Minimum Wavemaking Resistance

1. Introduction

Currently, computational fluid dynamics (CFD) is used as an analysis tool to study alternative ship hull form designs. Although extremely valuable, this approach suffers the limitation that it doesn't identify the optimum design. This is the motivation for developing CFD-based optimization methods wherein automatic determination of optimum shape is part of the simulation. One of the most important components of the CFD-based optimization is a geometry modeling method to provide a link between the design variables and a hull form. More specifically, the parametric expression and modification of the ship hull form is an essential feature for the automatic optimization.

A straightforward geometry modeling method is

superposition of several basic forms or expanding/ reducing basic geometry. This approach had been used for ship hull form optimization in combination with 3D potential flow solver¹⁰⁻⁶⁰ and with 3D Reynolds-averaged Navier-Stokes (RaNS) equation solver ⁷⁰⁻¹³⁰. More direct control of the form modification method was investigated by using partial differential equation or B-Spline method ¹⁴⁰⁻¹⁶⁰. Several capability studies on such geometry modeling were also done by others ^{170,180}.

On the other hand, a limitation of the abovementioned approaches is obvious, i.e., solutions of the optimization schemes are not directly linked with computer-aided design (CAD) model actually used by hull-form designers. Besides, modification trends proposed by the above-mentioned optimization schemes are sometimes far from those designers can readily accept. In order to solve those problems, it is clear that more advanced and capable approach must be introduced in the optimization scheme, i.e., implementation of the direct link with designer's CAD system for the CAD-based hull-form

^{*1} Read at the Spring Meeting of Kansai Society of Naval Architects, Japan, May 22, 2003, Received June 10, 2003

^{*2} Graduate School of Engineering, Osaka Prefecture University

^{*3} Mitsubishi Heavy Industries

^{*4} Denso Corporation

modification. The need of this approach was clearly stated for aerospace engineering applications ¹⁹⁾, and that must be a next challenge for application to ship hull-form optimization.

This paper concerns development of a CAD/CFD/ optimizer-integrated hull-form design system. The CAD, CFD, and optimizer modules are functionally independent and basically replaceable. Main objective of the present study is system development and demonstration of the capability, which justifies use of relatively simple flow and free-surface models, static sinkage and trim, and simplified design constraints. The CAD used in the present study is NAPA²⁰⁾, which is one of the most widely used CAD systems in the domestic as well as foreign shipyards. The CFD method is a Rankine-source panel method developed by one of the present authors. At present, two nonlinear optimization algorithms are available, i.e., the successive quadratic programming (SQP) and the genetic algorithm (GA); and results are presented for the former in the present paper. In the following, an overview of the present method is given, and results are presented and discussed for the bulbous bow optimization of a container ship for minimum In the discussion, the wavemaking resistance. present results are compared with those of others who conducted the similar optimization.

2. Overview of CAD/CFD/ Optimizer-Integrated Hull-Form Design System

2.1 General-Purpose Optimizer Module

Kernel of the present general-purpose optimizer module was developed in the precursory work ²¹⁾ and the details are discussed in the reference. Herein, an overview of the module is given. In use of this system, the module requires three settings to be given, i.e., (i) the number of design parameters, (ii) optimization algorithm, and (iii) definition of objective and constraint functions. The module is coded based on Visual Basic, and (i) through (iii) can be specified in spreadsheet screen. At present, SQP and GA are available for (ii). An important advantage of the present module is that one of two methods can be selected for evaluation of the objective and constraint functions, i.e., utilization of (a) EXE file, i.e., an executable file, and (b) DLL file.

Method (b) is useful if the objective and constraint functions are explicitly formulated. In the present study, method (a) is used as shown in Fig.1. The EXE file controls execution of CAD and CFD, and the related pre- and post-processing. This approach can be used with most of CAD and CFD codes, which are often Black Box for users.

The network-passage protocol is also implemented in the preset system. For example, the CFD method can be located and executed on a remote super computer. Capability of the approach was evaluated in the precursory work, and it appeared that CORBA is currently one of the most favorable interfaces for this application ²¹⁾.



Fig.1 Module interface using EXE files.

2.2 CAD Module (NAPA)

The CAD module used in the present work is NAPA system²⁰⁾, which is one of the most widely used CAD systems in the domestic as well as foreign As mentioned earlier, an important shipyards. feature for the automatic optimization is the parametric expression and modification of the ship hull form, which has been enabled by introduction of NAPA system in the present system. An example of the parametric expression using NAPA system is shown in Fig. 2. For the implementation, a consequential task is development of the linkage handling between the CAD and optimizer modules, and the CAD and CFD modules. In the present study, the NAPA Macro is used as shown in Fig. 3 for The procedures of hull form the purpose. CFD modification, geometry analysis, and pre-processing are described in the file. This approach is relatively straightforward but very effective to practical design application, which was also demonstrated and confirmed by others 22).



Fig.2 Parametric expression of ship hull form.



Fig.3 An example of NAPA Macro used in the present study.

2.3 CFD Module (Rankine-Source Panel Method)

A Rankine-source panel method (hereafter, referred to as R-S method) developed by one of the present authors 230 is used for the present study. Here, an overview of the method is given. First, the flow is assumed to be inviscid and incompressible, and its motion irrotational. Then, the total velocity potential is divided into two parts, i.e., the double model flow potential ϕ_0 and the free-surface flow potential ϕ . The former is known a priori if the ship geometry is given. In the present method, $\phi_{\rm b}$ and $\phi_{\rm l}$ are expressed by the Rankine type source distributed on the body surface S_b and undisturbed free-surface S_f, i.e.,

$$\phi_0 = \phi_\infty - \int_{S_b} \sigma_{0b} \left(\frac{1}{r}\right) dS \tag{1}$$

$$\phi_{1} = -\int_{b_{b}} \sigma_{1b} \left(\frac{1}{r}\right) dS - \int_{f} \sigma_{1f} \left(\frac{1}{r}\right) dS \qquad (2)$$

where ϕ_{∞} is the velocity potential of the uniform onset flow, σ_{ab} and σ_{ab} are source densities, and r is the distance between the field and singularity points. ϕ_0 and ϕ_1 are determined so as to satisfy the body boundary condition, i.e., $\partial \phi / \partial n = 0$, where is normal to the surface, and the linear n free-surface boundary condition, i.e.,

$$\phi_{ll}^{2}\phi_{lll} + 2\phi_{ll}\phi_{lll}\phi_{ll} + g\phi_{lz} = -\phi_{ll}^{2}\phi_{lll}$$
 $(z=0)$ (3)
where ℓ is the streamwise direction of double model
flow. The O-type free-surface panels are used for
free-surface boundary. Once the velocity potential is
determined, wavemaking resistance is obtained by
the surface integral of pressure field. Tahara ²³⁾

described more details of the numerical method.

f



Fig.4 General optimization procedure of the present system.

2.4 Integration of CAD/CFD/Optimizer Modules

In the following, an overview is given for overall in the present optimization procedure CAD/CFD/Optimizer integrated system. First, a general optimization procedure of the present system is summarized below and illustrated in Fig. 4.

- (1) Optimizer module is initialized with initial design parameters and starts optimization cycles.
- (2) CAD module modifies hull form, and at the same time, evaluates geometrical constraints, e.g., displacement.
- (3) CAD module generates surface data for CFD analysis.
- (4) CFD module reads surface data, and computes flow field and evaluates objective function, e.g., wavemaking resistance.
- (5) Optimizer module searches design parameters so as to decrease objective function under the given constraints.

-342-

(6) Repeat steps (2) through (6) until a sufficient convergence of objective function and design parameters is achieved.

The CAD and CFD modules are functionally independent and replaceable. For the present demonstration, the more specific system configuration is illustrated in Fig. 5. The SQP algorithm is used for the optimizer module. Comparison of capability between SQP and GA was performed in the precursory work and it was shown that the both methods yield satisfactory results and warrant further application to hull form optimization problem ²¹⁾. Details of the modules and the linkage handlings ① through ⑩ are described below.



Fig.5 Linkage handling between the modules.

- SQP is initialized with initial design parameters and starts optimization cycles.
- ② Design parameters are written in NAPA input file.
- ③ SQP executes CTRL.EXE and waits for completion of the CTRL. EXE procedure.
- ④ CTRL.EXE executes NAPA and waits for completion of the NAPA procedure.
- (5) NAPA reads design parameters and modifies hull form.
- (6) NAPA evaluates geometrical constraint functions and writes the values in NAPA output file. After writing the surface data in R·S input file, NAPA is terminated.
- ⑦ CTRL.EXE acknowledges termination of NAPA, then executes R-S method and waits for completion of the R-S procedure.
- (8) R-S method reads surface data, and computes flow field and objective function. After writing the data in output file, R-S method is terminated.

- ③ CTRL.EXE acknowledges termination of R-S method, then CTRL.EXE is terminated.
- ③ SQP acknowledges termination of CTRL.EXE, then reads constraint and objective function data from NAPA and R-S output files. SQP proceeds the next optimization cycle until a sufficient convergence of objective function and design parameters is achieved.

3. Optimization Test Case

3.1 Computational Conditions

As mentioned earlier, a main objective of the present study is system development and demonstration of the capability, which justifies use of static sinkage and trim, and simplified design constraints as described below. In the present study, the NAPA container ship is used as an initial hull form. Table 1 and Fig.6 show the principal particular and the hull geometry, respectively.

Table 1 Principal particular of NAPA container ship.



Fig.6 Initial ship hull form (NAPA container ship).



Fig.7 Locations of control points P1 and P2.

In the present study, the bulbous bow is optimized for minimization of wavemaking resistance at a given design speed. Three design parameters X_L , X_B , and X_H are defined as follows:

$$\begin{cases} X_L : X_L = L_{bub} / L_{PP} \\ X_B : X_B = B_{bub} / L_{PP} \\ X_H : X_H = H_{bub} / L_{PP} \end{cases}$$
(4)

where L_{bulb} , B_{bulb} , and H_{bulb} are the length, width, and height of bulbous bow, respectively. As shown in Fig. 7, the three design parameters are correlated to locations of control points P1 and P2. In the optimization, L_{PP} , B, and d are kept unchanged.

The objective function to be minimized is wavemaking resistance, which is computed by R-S method as described earlier. In the present study, it is assumed that total wavemaking resistance can be divided into two parts, i.e., the forebody and afterbody parts. In addition, the influences of bow modification on the latter are assumed small. This half body concept is often used in the design of container ship. Hence, in the present method, the following value is defined and used as the objective function to be minimized:

$$Cw_{f} = \frac{Rw_{f}}{1/2 \rho L_{PP}^{2} U^{2}}$$
(5)

where, Rw_T is the forebody part of the total wavemaking resistance. This approach will also avoid contamination of inaccurate information from the afterbody part where viscous effects on flow are significant and those are theoretically missing in the present method. In the computation, 1286 and 1200 panels are used for ship hull and free surfaces, respectively. See Fig. 8 for overview of the body and free-surface panel arrangement.



Fig.8 Body and free-surface panel arrangement for Rankine-source method (NAPA container ship).

In summary, the present optimization conditions are summarized as:

(a) Design speed : 22 knots (
$$Fn = 0.33$$
)
(b) Fix : Lpp, B, d
(c) Static sinkage and trim
(d) Initial design parameters :
 $(X_L, X_B, X_H) = (X_{L_Original}, X_{B_Original}, X_{H_Original})$
(6)

and optimization problem is defined as:

$$Cw_{f} \rightarrow Min.$$
Subject to:
(i) $|L_{bulb} - L_{bulb_Original}| \le 5\% L_{PP}$
(7)
(ii) $|B_{bulb} - B_{bulb_Original}| \le 10\% L_{PP}$
(iii) $|H_{bulb} - H_{bulb_Original}| \le 10\% L_{PP}$

Hence, three geometrical constraints (i) through (iii) are considered for the present demonstration.



Fig.9 Convergence history of design parameters and objective function in optimization cycles.



Fig.10 The initial and optimized hull forms (NAPA container ship optimization).



Fig.11 The initial and optimized hull forms (HTC container ship optimization)²⁴⁾.

-344-

3.2 Optimization Results and Discussion

Fig. 9 shows convergence history of the design parameters and the objective function. Variables are converged in 5 optimization cycles, and the objective function is decreased about 25%. Fig. 10 shows comparison of geometry between the original (initial) and optimal hull forms. For the optimal hull form, the lower part of the bulbous bow is more pronounced, and the volume distribution is increased in the region. Also, length of the bulbous bow is increased. For reference, the similar optimization results of others ²⁴⁾ are shown in Fig. 11. In the work ²⁴⁾, bow region of HTC (Hamburg Test Case) container ship was optimized using SQP and R-S method for minimum wavemaking resistance, and the hull form was modified by using a equation $y(x,z) = w(x,z) f_0(\xi,z)$, where w and f_0 are the modification function and the initial hull form, respectively. This hull-form modification is defined by 12 design parameters. The similar modification trends are observed between the two optimization results, which implies a validity of the present CAD based hull form modification method. It is also notable that the present CAD based method allows the use of smaller number of design parameters for the similar tends and magnitude of the hull form modification.



Fig.12 Comparison of wave profile between the original and optimal hull forms.

Fig. 12 shows comparison of wave profile between the present original and optimal hull forms. It is seen that differences in wave profiles are significant in the forebody, and those in afterbody are apparently negligible, which indicates a validity of the half body concept used in the present study. The bow wave crest is higher for the optimal hull forms, but the smaller elevation is indicated after the crest. The trends in wave profile are correlated with the local wavemaking resistance coefficient Cw_{see} as shown in Fig. 13. The Cw_{see} is defined by

$$Cw_{\rm sec} = \frac{Rw_{\rm sec}}{1/2\,\rho\,L_{PP}^{2}\,U^{2}}$$
(8)

i.e., Rw_{sec} is the wavemaking resistance evaluated by the surface integral between the two stations. Since the differences in the values between the original and optimal hulls are found negligible in afterbody, only forebody results are presented in the figure.



Fig.13 Comparison of local wavemaking resistance coefficient between the original and optimal hull forms.

The higher bow wave crest for the optimal hull is related to the larger cross sectional area in the bow region (see Fig.14 for the longitudinal distribution of cross-sectional area - Cp curve -). This causes increase in Cw_{sec} in the region, i.e., Bulb Top – S.S. 9.5. However, the wave elevation of the optimal hull form is lower after the crest, which results in the smaller Cw_{sec} in the region, i.e., S.S. 9.5 – S.S. 6.5. This is clearly attributed to lower objective function for the present optimal hull form since the differences in Cw_{sec} after S.S. 6.5 is negligibly small. It is notable that the bow wave crest of the optimal hull form will be smoothed if more strict design constraints are imposed, which was demonstrated by the present authors for the bow optimization of a naval combatant ²⁵⁾.





4. Conclusion

This paper concerns development of a CAD/CFD/ optimizer-integrated hull form design system. The CAD, CFD, and optimizer modules are functionally independent and basically replaceable. Main objective of the present study is system development and demonstration of the capability. Results are presented and discussed for the bulbous bow optimization of a container ship for minimum wavemaking resistance.

The present modification trends indicated in the results agree well with those shown in the similar optimization study presented by others. It is also notable that the present CAD-based geometry modeling method allows the use of smaller number of design parameters for nearly equal trends and magnitude of the hull form modification predicted by the conventional geometrical modeling method, which generally requires the larger number of design parameters.

Through the present demonstration, also including preliminary case studies which are not presented in this paper, it appeared that the present approach overcomes many issues related to unrealistic solutions often seen in the related studies, which is mainly due to the use of insufficient hull form modification methods. In conclusion, the present system has been shown very promising and warrants further investigation for more practical design constraints and conditions.

Acknowledgement

The present work has been partly supported by Grant-in-Aid for Scientific Research (2001-2003, Project Number 13555272; and 2002-2005, Project Number 14350524), and The U.S. Office of Naval Research under Award Number N0014-02-1-0256. The authors wish to thank whom concern the programs for their kind support and encouragement.

References

- Suzuki, K., Calisal, S.M., Tamashima, M.: Hull Form Improvement of Fishing Vessel by Non-protruding Bow Bulb, Naval Architecture and Ocean Engineering, Vol. 31, 1994, pp. 1-8.
- Lowe, T., Bloor, M., Wilson, M.: The Automatic Functional Design of Hull Surface Geometry, J. Ship Research, Vol. 38, No.4, 1994, pp. 319-328.
- Sakuma, S., Bessho, M.: A Theoretical Design for Auxiliary Ship Hull Forms, Proceedings of the 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design (OC'98), Osaka, Japan, 1998, pp. 407-423.
- Yasukawa, H.: Ship Form Improvement Using Genetic Algorithm, Ship Technology Research, Vol. 47, 2000, pp. 35-44.
- Masuda, S., Suzuki, K.: Experimental Verification of Optimized Hull Form Based on Rankine Source Method, J. Kansai Society of Naval Architects, No. 236, 2001, pp. 27-33 [Japanese].
- Dejhalla, R., Mrsa, Z., Vukovic, S.: Application of Genetic Algorithm for Ship Hull Form Optimization, Int. Ship Progress, 48, 2, 2001, pp. 117-133.
- Hamasaki, J., Himeno, Y., Tahara, Y.: Hull Form Optimization by Nonlinear Programming (Part 3) - Improvement of Stern Form for Minimizing Viscous Resistance -, J. Kansai Society of Naval Architects, No.225, 1996, pp. 1-6 [Japanese].
- 8) Tahara, Y., Himeno, Y., Tsukahara, T.: An Application of Computational Fluid Dynamics to

Tanker Hull Form Optimization Problem, Proceedings of the 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design (OC'98), Osaka, Japan, 1998, pp. 515-531.

- 9) Tahara, Y., Saitoh, Y., Himeno, Y.: CFD-Aided Optimization of Tanker Stern Form (1st Report)
 Minimization of Viscous Resistance -, J. Kansai Society of Naval Architects, No. 231, 1999, pp. 29-36 [Japanese].
- Tahara, Y., Saitoh, Y., Matsuyama, H., Himeno, Y.: CFD-Aided Optimization of Tanker Stern Form (2nd Report) - Minimization of Delivered Horse Power -, J. Kansai Society of Naval Architects, No. 232, 1999, pp. 9-17 [Japanese].
- 11) Tahara, Y., Ando, J., Himeno, Y.: CFD-Based Optimization of Tanker Stern Form Minimization of Delivered Horsepower Using Self-Propulsion Simulator -, Proceedings of the 8th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS 2001), Shanghai, 2001, pp. 719-724.
- 12) Hino, T., Kodama, Y., Hirata, N.: Hydrodynamic Shape Optimization of Ship Hull Forms Using CFD, Proceedings of the 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design (OC'98), Osaka, Japan, 1998, pp. 533-541.
- 13) Hino, T.: Shape Optimization of Practical Ship Hull Forms Using Navier-Stokes Analysis, Proceedings of the 7th International Conference on Numerical Ship Hydrodynamics, Nantes, France, 1999.
- Lowe, T., Bloor, M., Wilson, M.: The Automatic Functional Design of Hull Surface Geometry, J. Ship Research, Vol. 38, No.4, 1994, pp. 319-328.
- Huang, C.H., Chiang, C.C., Chou, S.K.: An Inverse Geometry Design Problem in Optimizing Hull Surface, J. Ship Research, Vol. 42, No.2, 1998, pp. 79-85.
- Peri, D., Rossetti, M., Campana, E.F.: Design Optimization of Ship Hulls via CFD Techniques, J. Ship Research, Vol. 45, No.2, 2001, pp. 1-10.
- 17) SR229 Annual Reports, Shipbuilding Research Association of Japan, 1996-1999 [Japanese, unpublished].

- Nowacki, H.: Hull Form Variation and Evaluation, J. Kansai Society of Naval Architects, No. 219, 1993, pp.173-184.
- 19) Samarreh, J.A.: Status and Future of Geometry Modeling and Grid Generation for Design and Optimization, J. of Aircraft, Vol. 36, No.1, 1999, 97-104.
- 20) Kuutti, I.: Impacts of Product Modelling on Ship Design and Production Planning, Proceedings of the 9th International Conference on Computer Applications in Shipbuilding (ICCAS), Yokohama, Japan, 1997; (also, further information is available at www.napa.fi or NAPA, NAPA Oy Ltd., FIN-00151, Helsinki, Finland).
- Sugimoto, S., Tahara, Y., Murayama, S., Saito, Y., Himeno, Y.: Development of User-Friendly Generalized Optimiza- tion System, J. Kansai Society Naval Architects, Japan, No. 238, 2002, pp.1-7 [Japanese].
- 22) Kobayashi, S., Mizutani, N., Nariyuki, E.: Automatic Hull Form Optimization System, NAPA User Meeting 2000, Helsinki, Finland, 2000.
- 23) Tahara, Y.: A Numerical Approach for Steady Ship-Wave Problem Based on Dawson-Type Rankine-Source Method with Non-H-Type-Topology Free-Surface Panels - 1st Report: with Main Emphasis on Application of O-Type-Topology -, J. Kansai Society of Naval Architects, Japan, No. 228, 1997, pp.79-90 [Japanese].
- 24) Yoshida, T., Suzuki, K.: Hydrodynamic Shape Optimization of Bow Bulbs Based on Rankine Source Method, J. Kansai Society of Naval Architects, Japan, No. 235, 2001, pp.39-47 [Japanese].
- 25) Tahara, Y., Paterson, E.G., Stern, F., Himeno, Y.: Flow- and Wave Field Optimization of Surface Combatants Using CFD-Based Optimization Methods, Proc. 23rd Symposium on Naval Hydrodynamics, Val de Reuil, France, 2000, CD-ROM.

マルチブロック NS/RaNS 法による アメリカ杯レース艇用風下帆走セールシステム 周りの流場解析

正員田原裕介*林豪**

Flow Analyses around Downwind-Sail System of an IACC Sailing Boat by a Multi-Block NS/RaNS Method

by Yusuke Tahara, Member Go Hayashi

Summary

The present study concerns analyses of flows around downwind-sail system of an International America's Cup Class (IACC) sailing boat. The numerical method is a multi-block Navier-Stokes and Reynolds-averaged Navier-Stokes equation solver (hereafter, referred to as multi-block NS/RaNS method) recently developed by the present author. The main objectives of the present work are two folds: (1) an initial capability evaluation of the present multi-block NS/RaNS method in analyses of large-scale-separation flow; and (2) application of the present method to flow analyses around downwind-sail system of IACC sailing boat, on which very few studies have been so far reported. In the present study, the mainsail/spinnaker configuration is particularly considered, where focuses are placed on the aerodynamic interactions between the mainsail and spinnaker as well as the influences of sail trimming on flow and aerodynamic forces. The present numerical method is based on an extension of the RaNS method for ship flow analyses for applications to more general fluid dynamic problems, which was successfully demonstrated through the present work.

1. 緒 言

スピンネーカーを用いたヨットの風下帆走の特徴は, セールシステム周りの大規模な剥離流場の生成とその 結果生じる抗力成分を推力に利用することである(実際 の帆走状態:Fig.1). これは風上帆走におけるセール推 力の生成と大きく異なる点であり,全体の流場構造や流 体力のセール間干渉などに関しては,これまでに明らか にされていない部分が多い.特に計算流体力学(CFD)を

* 大阪府立大学大学院

** トヨタ自動車(株)(研究当時 大阪府立学大学院工学 研究科)

原稿受理 平成15年7月10日

用いた研究については、風上帆走セールに関する事例は 比較的多いのに対し ^{い。},風下帆走セールに関するもの は殆ど報告されていない.

1



Fig.1 Downwind sailing of America's Cup boats (Auckland, 2000, Photo by Kaoru Soehata).

-348-

本研究は、近年著者らの研究グループで開発されたマ ルチブロック計算格子対応型ナビエスークス/レイノル ズ平均ナビエスークスソルバー(以下マルチブロック NS/RaNS 法), FLOWPACK Ver.2001a を用いた、アメ リカ杯レース艇用風下帆走セール周りの流場解析に関 するものである(アメリカ杯レース艇 · International America's Cup Class sailing boat:以下 IACC 艇と略称). 今回は特にメインセール・スピンネーカー構成を対象と する. それらのフライングシェープと帆走設定状態には, ニッポンチャレンジ・アメリカ杯2000技術委員会が 実際に計測したデータを用いた.本論文では、まず計算 手法の概要と基礎形状を対象とした精度評価の結果を 述べ,続いて上述した風下帆走セール周りの剥離流場の 詳細な検討だけでなく、メインセール・スピンネーカー の流体力学的相互干渉や、セールシステムの配置変化が 推力・横力に与える影響などについても詳細に議論する.

2









(Type C)

Fig.2 Matching types of multi-block boundaries.

2. 計算手法の概要

2.1 FLOWPACK Ver.2001a の概要

本研究で使用したマルチブロック NS/RaNS 法, FLOWPACK Ver.2001aの基礎部分は、従来著者らが船 体粘性流場への適用を主目的として開発してきた手法 (田原 5など)に基づいている. ここでは FLOWPACK Ver.2001aの概要を述べる. 流場の支配方程式は3次元 非定常,非圧縮流体に関する NS/RaNS 方程式である.数 値計算においては、支配方程式を数値的に生成した非直 交曲線座標系に変換し,輸送方程式を有限解析法により 離散化する.また速度場/圧力場結合には PISO タイプワ ンステップ法を用いている. FLOWPACK Ver.2001aの 従来法からの主な拡張点は、マルチブロック計算格子に 完全対応としたこと,乱流モデルとしてゼロ方程式モデ ル(Baldwin-Lomax モデル), 2 層 k- ϵ モデル, そして Blending k-ωモデルが採用でき、それらはスイッチ切替 によって自由に選択できることなどである. なお FLOWPACK Ver.2001a には自由表面計算機能も搭載 されているが、その詳細な報告は別の機会に行う予定で ある.特にマルチブロックへの対応は、並列高速計算へ の拡張を行う場合に極めて有用であり、その研究は現在 進行中である.

2.2 マルチブロック接合境界における流場情報の整合 方法について

マルチブロック法を適用する場合,各ブロックの接合 境界における流場情報の整合方法,すなわち情報を提供 するドナーブロックと,受取るレシピエントブロック間 の情報伝達の形態を定義する必要がある.FLOWPACK Ver.2001a では,Fig.2 に示すような3種の方法(Type A ~C)が採用できる.以下でその概要を述べる.

(Type A) Overlapping patched boundary:最も一般 的とされる整合方法であり、ドナーブロックとレシピエ ントブロックの境界面は、接合方向の格子一個分だけ重 複している.レシピエントブロックの境界条件はドナー ブロック内部の値によって直接与えられるため、整合方 法としては最も正確であると考えられる.Fig.2 (Type A)の表記に従うと、各ブロックの境界面における流場情 報 f_{A1} および f_{B1} の交換は以下のように行われる.

$$\begin{cases} f_{A,1} = f_{B,2} \\ f_{B,1} = f_{A,2} \end{cases}$$
(1)

(Type B) Non-overlapping patched boundary:上述 した Type A に類似するが,格子の重複(overlap)を用い ない方法である.接合面における流場情報を,各ブロッ クの内部の値と適当な補間式を用いて与える. Type A より簡便であるため比較的多くの計算事例で用いられ ており,本研究においても基本的にはこの方法を採用し ている.現行の補間式は,Fig.2(Type B)の表記に従い格 子間隔を Δ としたときに以下のようになる.

$$f_{A,1} = f_{B,1} = \frac{\Delta I_B \cdot f_{A,2} + \Delta I_A \cdot f_{B,2}}{\Delta I_A + \Delta I_B}$$
(2)

-349-

(Type C) Overlapping non-patched boundary:上述 した Type A および B の方法は,原則的に整合面におけ る計算座標2方向の格子を整合配置(patched)すること を前提としている.一方 FLOWPACK Ver.2001a では, 特定のブロック内でより多くの格子を用いたい場合な どに対応できるように,ここで述べる Type C が適用で きる. Fig.2 (Type C)に示すように,レシピエントブロ ックの境界面をドナーブロックの内部に配置する必要 があるが,上述した格子の整合配置の必要はない.結局, レシピエントブロックの境界面における流場情報 $f_{A,1}$ および $f_{B,1}$ をドナーブロック内部の流場情報 $f_{A,1}$ および $f_{B,1}$ をドナーブロック内部の流場情報 $f_{A,1}$ および $f_{B,1}$ をドナーブロック内部の流場情報 $f_{A,1}$ および $f_{B,1}$ をドナーブロック内部の流場情報 $f_{A,1}$

$$\begin{cases} f_{A,1}(x,y,z) = \sum_{i=1}^{2} \sum_{j=1}^{2} \sum_{k=1}^{2} \alpha_{ijk} f_{B,ijk} x^{i-1} y^{j-1} z^{k-1} \\ f_{B,1}(x,y,z) = \sum_{i=1}^{2} \sum_{j=1}^{2} \sum_{k=1}^{2} \beta_{ijk} f_{A,ijk} x^{i-1} y^{j-1} z^{k-1} \end{cases}$$
(3)

2.3 マルチプロック計算格子の品質評価法および不確 かさ解析

一般に、物理空間のマルチブロック分割が進むに従い、 個々のブロックにおける格子品質の維持が困難になり やすい.単に1個のブロックで格子品質が低下すれば、 それは全体的な解の信頼性の低下に発展する可能性が ある.本研究では、比較的簡単な評価パラメータを用い、 上述の問題を軽減する手法を検討した.評価パラメータ には次式で定義されるJ^{*}を採用した.

$$J^{*} = \frac{J}{|\mathbf{a}_{1}| \cdot |\mathbf{a}_{2}| \cdot |\mathbf{a}_{3}|} = \sqrt{\frac{g}{g^{11} \cdot g^{22} \cdot g^{33}}}$$
(4)

ここで

$$\begin{cases} \mathbf{a}_{1} = (x_{\xi}, y_{\xi}, z_{\xi}) \\ \mathbf{a}_{2} = (x_{\eta}, y_{\eta}, z_{\eta}) \\ \mathbf{a}_{3} = (x_{\xi}, y_{\xi}, z_{\xi}) \end{cases}$$
(5)
$$\begin{cases} g^{11} = |\mathbf{a}_{1}|^{2}, \ g^{22} = |\mathbf{a}_{2}|^{2}, \ g^{33} = |\mathbf{a}_{3}|^{2} \\ g = [\mathbf{a}_{1} \cdot (\mathbf{a}_{2} \times \mathbf{a}_{3})]^{2}, \ J = \sqrt{g} \end{cases}$$
(6)

上式中の(ξ,η,s)は計算空間で定義される計算座標系で ある. $J^*(0 < J^* \le 1)$ は座標変換パラメータとして計算さ れるヤコビアンJの無次元値であり,定義セルが直方体 形状において1となり,格子歪が拡大するにしたがって 減少する.幾何学的にはセル体積量を表すヤコビアンを 上式のように無次元化することにより,格子歪の傾向の みを抽出することができる.本研究では、実際にマルチ ブロック格子の生成過程でこのパラメータを用い、適宜 3D等値面を表示する画像処理と組み合わせることに よって,効率的に高品質の格子が生成できることを確認 した.

一方, 文献 ^{6),7)}の方法に基づく数値計算不確かさの評 価も行った.後述する全ての計算例(円盤・矩形平板・円 錐・およびダウンウインドセール)において精度値 p=1.5-2.5 は従来の計算例より推定可能であると仮定 し,基本格子を基準に3方向格子改善率 n=2として評 価した結果, 流体力に関する Simulation Uncertainty Uswは2.5.6.3%であった. すなわち, 数値計算のノイズ として捉えられるべき幅は最大約 6%であり、複数の計 算値の有意な差を論じる場合は、その差の絶対値は Usw 以上である必要がある.この Uswは船体流場を解く近年 の CFD の観点では比較的高いものであるが、今回対象 とした流場の複雑さを考慮すれば、ある程度妥当なレベ ルであると思われる.一方,本論で論じる全ての計算例 において、比較に用いた参考値の不確かさ Upが不明で あったため、文献 6,7で提案されている Validation Uncertainty Uvの評価は行っていない.

3

3. 計算結果-基礎形状を用いた精度評価

ここでは比較的簡単な形状を対象とした計算を行い、 本計算手法の精度確認を行った結果について述べる.本 研究で対象とする流場の特徴は、大規模な剥離を有する こと,そしてその流体力への影響,すなわち圧力成分が 支配的となることである. そのような流場をもたらす基 礎形状として、流れに対し垂直に置かれた円盤(disk)と 矩形平板(square plate),そして頂点を上流方向に向け て設置した円錐(60 deg. cone)を対象とした計算を行っ た. またそれらの抵抗参考値には White®が示すものを 用いた、一方、ここで対象とした形状は比較的簡単なも のであるが、NS/RaNS 法で流場を解析する際にはマル チブロック格子の適用が最も有効である.今回の計算に おいては、円盤、矩形平板、そして円錐のそれぞれに関 し4ブロック、6ブロック、そして3ブロックのマルチ ブロック格子を用いた. 設定レイノルズ数は White®の 抵抗参考値に従って Rn= 1.0×104 とし、計算格子数は 約10万点とした. なお以下で検討する計算結果は一様 流,流体密度,そして代表長(円盤の場合は円盤直径)で 無次元化し,また抵抗値の場合は投影面積を用いて無次 元化している.

まず乱流モデルを使わない NS 方程式の解を検討する. 本研究の手法を用い,上述した3つの基礎形状について NS 方程式の解を求め,その抵抗値と White[®]の抵抗参 考値との比較を Fig.3 に示す.White[®]の値はあくまで 参考値であるために定量的な精度の議論は困難である が,その抵抗参考値が示す傾向はほぼ正確に予測できる ことが示されている.

-350-




4





Fig.4 Comparison of iso-velocity (*U*) surfaces between NS and RaNS solutions (Disk plate, $Rn=10^4$).



Fig.5 Comparison of drag coefficient between NS and RaNS computations (Disk plate, *Rn*=10⁴).



Fig.6 Comparison of drag coefficient between NS and RaNS computations (T-10 Parachute form, $Rn = 3 \times 10^6$, Sahu et al.⁹⁾).

次に乱流モデルを使用した場合を検討する.上述した ものと同一の円盤に関し,流場を NS 方程式および RaNS 方程式(Baldwin Lomax モデルを使用)を解いて 求めた主流方向速度の等値面図を Fig.4 に示す.同図よ り,円盤後方に存在するバブル型剥離領域の概況を観察 することができる.NS および RaNS の結果ともにほぼ 同様な規模の逆流域を示すが,RaNS 解は NS 解に比べ やや早い速度回復を示している.一方,Fig.5 に示す抵 抗値の比較では,RaNS 解の方が NS 解より若干大きい ものの両者の解はほぼ同等であることが示されている.

今回のものと類似した流場で RaNS 法の従来型乱流 モデルを検証した事例として,パラシュート周りの流場 に関する Sahu ら⁹⁰の研究がある.Sahu らは同一の計算 コードを用い,開傘後の定常降下状態(レイノルズ数約 3.0×10^{6})にある供試形状(文献ではT-10と定義)に関し, NS 方程式の解,および1方程式ならびに2方程式乱流 モデルによる RaNS 方程式の解を比較した.設定レイノ ルズ数は約 3.0×10^{6} とされている. Fig.6 にその抵抗値 の比較を示す.その結果,抵抗計測値と計算値の差は NS 解, RaNS 解(1 eq. model), RaNS 解(2eq. model) でそれぞれ-1.2%, -2.5%, -5%であり,NS 方程式の解が 最も計測値に近かったことを報告している. Sahu らは Smagorinsky モデルを用いた計算(Fig.6 中 LES)も行っ たが,その結果と実験値との差は約+12.7%となり,NS や RaNS の結果を改善するには至らなかった.

上述した円盤に関する本研究の結果や Sahu らの結果 を同時に考察する限り,今回対象とした流場,すなわち 大規模な剥離が存在し,かつ流れの剥離地点がほぼ固定 しており,さらに圧力成分が支配的となる場合の流体力 の予測という観点では,従来型乱流モデルの有効性がや や曖昧であると同時に,NS 計算によってもある程度妥 当な結果が得られるということがいえる.また従来型乱 流モデルは壁近傍の格子に関する制約が強く(研究事例

-351-

として文献 ¹⁰など), 乱流モデルを用いない NS 計算に よっても妥当な精度が確保できるのであれば, 実用上極 めて有用であるとも考えられる.



Fig.7 Circulation around mainsail and spinnaker.

4. 計算結果-風下帆走セール

4.1 IACC 紙風下帆走セールシステムの従来型設計法に おける問題点および一般的な流力特性について

具体的な流場解析を議論する前に、ここでは特に IACC 艇の風下帆走セールの従来型設計法で指摘されて きた問題点と一般的な流力特性を概説する. Fig.1 に示 すように、現 IACC 艇の風下帆走セールシステムは、船 体中央部近傍のメインセールと船体前方に張出したス ピンネーカーなどの2枚のセールで構成されている.ス ピンネーカーは基本的に左右対称であるが、これに代わ ってジェネカーと呼ばれる非対称セールを用いること もある. 通常メインセールは風上・風下帆走ともに同一 のものを使用するのに対し,スピンネーカーは風下帆走 に特化して使用する.スピンネーカーのセール面積はメ インセールよりはるかに大きく、よって推力の大半はス ピンネーカーで生成されるとともに、メインセールの機 能としてはスピンネーカー風上側の流場の整流効果が 重要であると考えられている.軽材質で形成されるスピ ンネーカーは張力制御によって形状をコントロールし やすい反面、近傍流場の状態によって容易に形状崩壊が おき、それによって著しい推力低下をもたらすことも知 られている.

既述したとおり,スピンネーカーを用いたヨットの風 下帆走の特徴は,セール周りの大規模な剥離流場の生成 と,その結果生じる抗力成分を推力に利用することであ る.これは風上帆走におけるセール推力の生成と大きく 異なる点であり,全体の流場構造や流体力のセール間干 渉などに関しては,これまでに明らかにされていない部 分が多かった.その理由としては,スケールエフェクト が著しいために模型実験による検討が困難であるだけ でなく,実船帆走試験においても流場や流体力の計測が 難しいこと,加えて風上帆走セールの設計で比較的多用 されるパネル法などの非粘性流法では,大規模な剥離流 場の効果を正確に表現できないことなどが挙げられる. これらの理由により,従来型設計法では設計者やセール トリマーの経験に基づく手法が主流となっている.

5

一方,実際のIACC 艇の風下帆走では,セールシステ ムが生成する揚力成分の一部も推力として利用してお り,現場で観測される現象には揚力体干渉による効果と 考えられるものが報告されている.例えば,スピンネー カーの使用によるメインセール揚力の減少や、メインセ ールの使用によるスピンネーカー有効迎角の増加など はその具体例と考えられ、それらを循環モデルによって 図示すると Fig.7 のようになる. 揚力体干渉時の循環強 さを単体時のものと比較した場合、メインセール循環 Γ_мは減少し,スピンネーカー循環 Γ₆は増加すると考え られ、これによって上述した実現象を大まかに説明でき る. またこれらの現象は、風上帆走時の2枚セールの干 渉に関する検討 2,4によって報告されたものと類似した 効果である.しかしながら,流場の剥離現象を含む顕著 な流体粘性影響を考慮した考察を行うためには、上述し た循環モデルだけでは明らかに不十分であると考えら れる.

以上の考察の結果,本研究では特に大規模な剥離を伴う複数揚力体まわりの流場特性に着目し,次の2つの観点:剥離流場の効果を正確に表現できる本研究の流場解 析手法の適用による「(i)メインセール・スピンネーカー の流体力学的相互干渉」,および「(ii)各セールの配置変 化が他方のセールおよびセールシステム全体の流体力 と流場へ与える影響」の検討に焦点を絞り,実際に観測 されている現象との比較も含めながら計算結果を議論 する.

4.2 メインセール・スピンネーカー構成に対するマルチ ブロック計算格子および計算条件の概要

本計算手法の適用において、まず Fig.8(a)に示すよう な座標系を定義する.同図には流体力の方向(Fig.8(b)) やセール配置に関する角度の定義(Fig.8(c))も示してい る.続いてスピンネーカーとメインセールに関し、Fig.9 に示すような CAD モデルを作成した.そのセール形状 には実測で得られたフライングシェープを用いた.本研 究ではセールはハードセールとし、またその厚みは無視 できると仮定、加えて船体の水面上の形状・マストなど の付加的構造物の影響は考慮せず、さらに海面はフラッ トな鏡像面として扱う.この CAD モデルのセール部分 周囲の流場を5ブロックで分割し、Fig.10 に示すマルチ ブロック計算格子を生成した.計算効率と精度の観点よ

-352-

り、本研究で最終的に用いた総格子数は約 29 万点であ る.格子生成の際には、既述した格子品質評価パラメー $タJ^* を適宜確認する方法を用いた.一例として、Fig.11$ $には改良前の格子の<math>J^*$ -3D等値面を示す.改良前の格 子にはマストトップ近傍に水平方向に広がる格子歪が 著しい部分(図中 $J^*=0.5$ および 0.4)が検出されている. この情報に基づいた格子改良の結果を確認するために、 改良前後のマストトップ水平面断面内の J^* 分布を Fig.12に示す.改良前の格子において図中Aで示されて いた低値部分は、改良格子では明らかに消滅しているこ とが確認できる.







Fig.9 CAD-based surface modeling of the present mainsail/spinnaker configuration.

また,前節で議論した乱流モデルの必要性に関する検 討結果に基づき,本流場解析の計算条件が実スケールで Sahuら®の結果と近い10⁶レベルのオーダーであること, そして対象とする流場がほぼ固定した剥離地点と大規

-353-

模な剥離領域を有し、かつその流体力への影響が支配的 となること、さらに本流場解析の目的が流場および流体 力の基礎的特徴の理解を目的としていることを主な理 由として、今回は乱流モデルを使用しないことに決定し た.実際に以下で示す全ての計算結果において、流体力 に占める表面摩擦応力成分は圧力成分よりも2オーダ ー小さいことを確認している.これは船体境界層などを 解析する場合と大きく異なる点であり、部分的な乱流遷 移などの乱流効果が流体力に与える影響は無視できる と考えられる.



Fig.10 Multi-block arrangement for the present computational grid.









6

一方,計算における境界条件は主流方向(X 軸正方向) に一様流が流入するとし、Fig.10の表記による流入境界 SI, 側面境界 Ss, および上面境界 ST において一様流条 件を, また海面境界 SB においては鏡像面条件を課す. さらに流出境界 SE においては伴流中の速度場の主流方 向ゼロ拡散条件およびおよび圧力場の主流方向ゼロ勾 配条件に基づく境界条件を課す.FLOWPACK Ver.2001a では、これらの境界条件もマルチブロック接 合面における設定と同様,入力コントロールファイルに よって容易に設定することができる. なお以下の議論で 示す計算値の長さスケールはマスト高さで無次元化し, また流速と圧力は流体の密度と一様流速で、さらにメイ ンセールやスピンネーカーの個別の流体力に関しては それぞれの面積を用いて(それぞれ実スケールで約 200m² および 450m²), ならびにセールシステム全体の 流体力はそれら合計の面積を用いて無次元化している.

Table 1 Sail system configuration of single- and

two san	conuntions.			
Trim	case 1	case2	case 3	
condision	Spin	Main Main +Spin		
Main Attack Angle(deg)		56.0	56.0	
Spin Attack Angle(deg)	41.5		41.5	
			R	



Fig.13 Overview of flow field around the present mainsail/spinnaker configuration (Case 3 in Table 1).

4.3 メインセール・スピンネーカーの流体力学的相互干 渉について

ここでは特にメインセール・スピンネーカーの流体力 学的相互干渉を検討した結果について述べる. 帆走状態 とセールシステム配置を Table 1 に示す. そこに示すよ うに、スピンネーカー単体(Spin)、メインセール単体 (Main)、そして両者同時配置(Main + Spin)の3状態を 比較することにより標記の検討を行った.なお Table 1 の設定は、真風速 12 ノットの場合に実測された日本艇 の特定帆走状態に基づいている.見かけ風向(AWA)は 111°であり、またそのときの見かけ風速(AWS)は 4.4 ノットであった.なおこの状態のマスト高さ(33m)に基 づくレイノルズ数(Rn)は約 5.23×10⁶となり、この実ス ケール状態で計算は行っている.Table 1 (Case 3)の設 定における計算結果の3次元流線とセールスパン中央 断面内の主流方向速度分布の様子を Fig.13 に示す.ス ピンネーカー下流域には大規模な剥離流場が形成され、 またその流場はセール間相互干渉の強い影響下にある ことが予想できる.

7

まずメインセール流体力に及ぼすスピンネーカーの 影響を検討する、メインセール単体状態とメインセー ル・スピンネーカー同時配置状態において、メインセー ルに生じる流体力の計算結果を Fig.14(a)に示す. スピン ネーカーの存在によりメインセール流体力は平均約 17%減少していることが分かる.既述したとおり、この メインセール流体力の減少は、実際のセーリングにおい てセーラーにより認識されていた現象と合致するもの である.この現象を考察するために、水平断面速度分布、 圧力分布,そして水平断面流線を詳細に調査した.その 一例としてメインセール・スパン中央水平断面における 解を Fig.15 に示す. スピンネーカーの存在によりメイ ンセールの前縁と後縁の速度が減少しているが、これは Fig.7 に示す揚力体間干渉の効果によってメインセール 循環 Γ_M が単体時より減少したためであると考えられ る. 同時にメインセール風下側の流場が整流され、剥離 領域が減少するとともにその領域の圧力が総じて上昇 している. これらの効果は、全て推力をはじめとするメ インセール流体力を減少させる方向に作用している.

つぎにスピンネーカーに関しても同様な検討を行う. スピンネーカー単体状態とメインセール・スピンネーカ ー同時配置状態において,スピンネーカーに生じる流体 力を Fig.14(b)に示す.今度はメインセールの存在によっ てスピンネーカーに生じる流体力が平均 8.2%増加して いる. このスピンネーカー流体力の増加も実現象と合致 する効果である.まずスピンネーカー周りの流線に着目 すると、メインセールの存在によってスピンネーカー後 緑下流の剥離領域が拡大していることが分かる.同時に その拡大した剥離領域でさらなる圧力低下が起こり (Fig.16 のスパン中央部の圧力分布も参照),これにより スピンネーカーに生じる推力などの全ての流体力が増 加している.

-354-

日本造船学会論文集 第194号

8







Fig.15 Comparison of velocity magnitude and pressure contours and cross-sectional streamlines (Mainsail mid-span section): (a) Spinnaker only; (b) Mainsail only; and (c) Mainsail + Spinnaker; rows, velocity magnitude contours, pressure contours, and streamlines, respectively.



Fig.16 Comparison of pressure profiles between single (spinnaker) and two-sail conditions at the spinnaker mid-span section.



(a) Spinnaker only

(b) Mainsail + Spinnaker

Fig.17 Comparison of streamlines near the spinnaker windward edge at the mid-span section.



(a) Spinnaker only
 (b) Mainsail + Spinnaker
 Fig.18 Comparison of pressure contours near the spinnaker windward edge at the mid-span section.

これらの現象は、Fig.7 の循環モデルにおいてはメイ ンセール循環 Γ_M の影響によってスピンネーカー循環 Γ_S がその単体時より増加し、それに伴いスピンネーカ ー前縁近傍の流れの流入迎角よび速度が増加したため であり、特にその流入迎角の増加は Fig.17 に示すスピ ンネーカー前縁近傍の流線によっても確認できる.一方、 Fig.18 に示すスピンネーカー前縁近傍の圧力分布に着 目すると、エッジ近傍下流側において単体時より圧力が 低下していることが確認できる.これはその領域におけ るスピンネーカーの形状崩壊を抑止する方向に作用し, この傾向も実際のセーリングにおいて観測されている 効果と合致するものである.

9

一方,上述した流体力の特に推力成分に占める抗力成 分の寄与を調べた結果,ここで設定したメインセールと スピンネーカー両者同時配置状態において,それぞれの セール推力に占める抗力寄与は約29%と約26%であり, またセールシステム全体の推力に占める抗力寄与は約 27%であった.この抗力寄与の割合は,より風下側へ進 路をとった帆走状態ではさらに拡大することが予想で きる.これはここで対象としたような流場における流体 力の正確な推定には,剥離現象をはじめとする顕著な流 体粘性影響を考慮できる流場解析手法の適用が不可欠 であることを裏付ける結果であると考えられる.

 Table 2
 Computational test cases: varied spinnaker

 trim angles for fixed mainsail angle.

Main Attack Angle(deg)	56.0	56.0	56.0	56.0	56.0	56.0
Spin Attack Angle(deg)	33.4	37.7	41.5	45.2	48.8	55.4
Horizontal section z/Span=50.1%	A	A	A	R	R	Â

 Table 3
 Computational test cases: varied mainsail

 trim angles for fixed spinnaker angle.



4.4 セールシステムの配置変化が推力・横力に与える影響について

ここでは前節の議論では固定していたメインセール およびスピンネーカーの配置角度を変化させた場合に, セールシステムが生成する流体力の変化を考察する.風 の流入条件の設定やレイノルズ数も同条件とする.以下 で検討するスピンネーカーおよびメインセールの配置 を Table 2 と Table 3 に示す.角度の定義については Fig.8 を参照されたい.なおここで定義するセール角度 が増加すれば,幾何学的なセール迎角も増加する.

まず Table 2 に示すようにメインセールの設定角度を 固定し,スピンネーカーの角度を変化させた場合を考察 する.スピンネーカーの角度変化に対するメインセール, スピンネーカー,そしてセールシステム全体の推力およ

び横力変化を Fig.19 に示す. スピンネーカーの設定角 度が増加するに従い、メインセール推力は単調減少を示 すのに対し、スピンネーカー推力は49°付近で最大とな る傾向を示す.前節と同様な方法で流場を調査した結果. スピンネーカー角度が増加するに従い、前節でも議論し たメインセール下流側の整流効果がより顕著になり、剥 離領域が一様に減少することによってメインセール流 体力が減少し, それに伴いメインセール推力および横力 がともに減少したことが分かった. またスピンネーカー に関しては、スピンネーカー角度が増加するに従いその 下流の剥離領域は拡大していき,スピンネーカーが生成 する流体力自体は増加し、それにともなって横力成分も 増加することが確認できた.一方、ヨットの進行方向成 分としてのセールシステム全体の合計推力は、スピンネ ーカー角度 41°付近で最大値をとる傾向が示されてお り、これより小さい角度ではスピンネーカー前縁近傍下 流側の表面圧力が上昇し、具体的には基準圧(Cp=0)を上 回る部分が生じることを確認した. 前節でも述べたよう に,この圧力上昇はスピンネーカー前縁形状の崩壊に関 連すると考えられる、実際のセーリングにおいては、ス ピンネーカー前縁形状が崩壊する直前までスピンネー カー角度を減少させ、セールシステム全体の推力を最大 にするという操作が経験的に行われるが、本研究の結果 はその操作の意味を流場の詳細な解析によって理論的 に裏付けたものであると考えられる.

つぎに Table 3 に示すようにメインセール角度を変化 させた場合を考察する.スピンネーカー角度は上述した 設定でセールシステム全体の最大推力を得た角度に固 定した.メインセールの角度変化に対するメインセール, スピンネーカー、そしてセールシステム全体の推力およ び横力の変化を Fig.20 に示す.まず前節と同様に流場 を調査した結果、特にメインセール推力が最大となる角 度以上においてはメインセール下流の剥離領域が急速 に拡大し、従来このセールの重要な目的とされている当 該流域の整流効果が急減することが分かった. なお今回 の検討の範囲では、メインセール角度の変化はスピンネ ーカー下流の剥離領域の規模には顕著な変化をもたら さず、またスピンネーカー流体力に与える影響は比較的 小さいことが分かった. これは Fig.7 に示す揚力体相互 干渉の観点で考えると、面積が相対的に小さくかつ後方 に位置する揚力体(メインセール)の循環変化が他方の揚 力体(スピンネーカー)の循環変化に及ぼす影響は、その 逆方向の影響に比べて小さいことにも関連すると考え られる.

ー方メインセールの角度の変化によって,メインセー ル流体力自体は明らかに変化する.まず横力変化に着目 すると、メインセールの角度が増加するに従い、メイン セール横力はほぼ線形的に増加しており、これは上述し たメインセール下流の剥離領域の拡大に関連する効果 である.一方、ヨットの進行方向成分であるメインセー ル推力はメインセール角度が56°付近で最大となるが、 セールシステムの合計推力は61°付近で最大値をとる 傾向が示されている.実際のセーリングにおいてはメイ ンセールを若干オーバートリム状態にし、セール背面の 剥離領域の拡大をある程度許容するように設定するこ とによってセールシステム全体の最大推力を求めると いう操作が行われる.この経験的な操作に関しても、本 研究の結果はその操作の意味を理論的に説明するもの であると考える.







Fig.20 Influences of mainsail angle on sail-system aerodynamic forces (for fixed spinnaker angle).

-357 -

今回行った流場解析の重要な意義の一つは、従来明確 にされていなかった流場と流体力の関連が明らかにで きるとともに、Fig.21に示すようなセール表面の流体力 分布が容易に得られることである.Fig.21は今回の検討 でセールシステムが最大推力を示したスピンネーカー およびメインセールの設定角度における推力および横 力分布をセール表面で表示したものである.なおここで 得られたセール設定角度に関する傾向は、セーラーが報 告する経験値と良く一致するものであった.本研究で得 られたような詳細な流場や流体力に関する情報は、セー ル設計において極めて有用であるとともに、最適セーリ ングに関する情報をセーラーに提供することを可能と し、それによってより理論的にヨットの帆走性能および セーリング技術の向上を図ることが可能になると考え られる.



(Thrust) (Side force) Fig.21 Comparison of thrust and side force distributions (Mainsail angle 61.0 deg. and Spinnaker angle 41.5 deg.).

5. 結 言

本研究では、近年著者らが開発したマルチブロック NS/RaNS 法-FLOWPACK Ver.2001a-をアメリカ杯レ ース艇の風下帆走セールシステムに適用し、主にメイン セール・スピンネーカー構成に関する流場解析を行った. より具体的には、本セール構成に関し従来不明確とされ てきた剥離流場の詳細な検討だけでなく、メインセー ル・スピンネーカーの流体力学的相互干渉や、セールシ ステムの配置変化が推力・横力に与える影響などを調査 することを目的とした.

本研究では、まず基礎形状を対象とした計算手法の精

度評価を行い,本研究で主眼とした大規模剥離流場の解 析と流体力の評価という観点において,本手法は妥当な 精度を有することを示した.加えて乱流モデルの有効性 に関する検討も行った結果,今回対象とした流場,すな わち大規模な剥離が存在し,かつ流れの剥離地点がほぼ 固定しており,さらに圧力成分が支配的となる場合の流 体力の予測においては,乱流モデルを使用しない計算に よっても妥当な結果が得られるという見解に至り,これ をセールシステムの流場解析にも適用した.

本研究の計算手法を用いてメインセール・スピンネー カーの流体力学的相互干渉を検討した結果,その流場の 解釈には従来風上帆走セールにおいて有効とされてき た2枚セールの揚力体間干渉に基づく循環モデルもあ る程度有用であるが,それに加えて流場の剥離現象を含 む顕著な流体粘性影響を考慮した考察が必要であり,そ の目的において本研究の計算手法は極めて有効である ことを示すことができた.それは今回の流場解析で得ら れた流場や流体力の傾向が,実船帆走試験で経験的に認 識されていた多くの現象を説明できたことによっても 裏付けられると考える.加えてセールシステムの配置変 化が推力・横力に与える影響に関しても,多くの観点で 従来セーラーが経験的に認識していたものと同じ傾向 が予測できたとともに,それらの操作の意味をより理論 的に説明することが可能となった.

今回行った流場解析の重要な意義の一つは、従来明確 にされていなかった流場と流体力の関連が明らかにで きたとともに、風下帆走セールシステムに関する多くの 詳細な流場・流体力情報を提供できる可能性を示したこ とである.これらはセール設計において極めて有用であ るとともに、最適セーリングに関する情報をセーラーに 提供することを可能とし、それによってより理論的にヨ ットの帆走性能およびセーリング技術の向上を図るこ とが可能になると考えられる.

なお本計算手法の今回の適用を通じ,従来船体粘性流 場の解析を目的に開発されてきた NS/RaNS 法が,完全 マルチブロック格子対応型へ拡張されたことによって, より一般的な流体工学問題に適用できる可能性を示す ことができた.これも本研究の重要な成果の一つである と考える.

왦 緒

本研究を行うにあたり種々のご助言ご協力をいただ きました大阪府立大学大学院工学研究科・姫野洋司先生, 同・勝井辰博先生,ならびに有限会社エイ・シー・ティ ー・鹿取正信氏,同・金井亮浩氏に感謝の意を表します. なお本研究の一部には文部科学省研究費補助金 (2002-2005, Project Number 14350524)の援助を受け たことを記し、関係各位に感謝いたします.

▶ 考 文 献

- Fukasawa, T. and Katori, M.: Numerical Approach to Aeroelastic Responses of Three-Dimensional Flexible Sails, Proc. The 11th Chesapeake Sailing Yacht Symposium, SNAME, Chesapeake, 1993, pp. 87-105.
- Lee, Y.W., Miyata, H: Application of CFD Simulation to Design of Sail, J. Marine Science and Technology, Vol. 4,1999, pp. 163-172.
- 3) 西川達雄,田原裕介,正岡孝治,姫野洋司:セールを 対象とした粘性流対中における空力弾性問題(第 2報)-繊維材質の変更,制御方法の改善につい て-,関西造船協会誌,第232号,1999,pp.39-46.
- 高野浩太郎,宮田秀明:セール CFD 解析による AC 艇帆走性能評価,関西造船協会誌,第 237 号, 2002, pp. 27-32.
- 5) Tahara Y.: A Multi-Domain Method for Calculating Boundary-Layer and Wake Flows around IACC Sailing Yacht, 関西造船協会誌, 第 226 号, 1995, pp. 63-76.
- Stern, F., Wilson, R.V., Coleman, H.W., Paterson, E.G.: Comprehensive Approach to Verification and Validation of CFD Simulations - Part 1: Methodology and Procedures, ASME J. Fluid Engineering, Vol. 123, 2001, pp. 793-802.
- Wilson, R.V., Stern, F., Coleman, H.W., Paterson, E.G.: Comprehensive Approach to Verification and Validation of CFD Simulations • Part 2: Application for RANS Simulation of a Cargo/Container Ship, ASME J. Fluid Engineering, Vol. 123, 2001, pp. 803-810.
- White F.M.: Viscous Fluid Flow, McGraw Hill, New York, NY, 1991.
- 9) Sahu, J., Edge, H., Heavey, K., Stein, K., Benney, R., Chakravarthy, S.: Comparison of Numerical Flow Field Predictions for Army Airdrop Systems, Proc. 15th CEAS/AIAA Aerodynamic Decelerator Systems Technology Conference, Toulouse, France, 1999, pp. 110-120 (AIAA-991715).
- 10) Tahara, Y., Katsui, T., Himeno, Y.: Computation of Ship Viscous Flow at Full

Scale Reynolds Number · Consideration of Near-Wall Flow Model Including Surface Roughness Effects, 日本造船学会論文集, 第 192 号, 2002, pp. 89·101.

-359-

関西造船協会論文集 第241号 平成16年3月

PCクラスター並列計算環境における大規模・高効率CFDの

コーディング技術の開発*1

-第1報:MPIプロトコルを適用したコード開発環境の構築と初期評価-

正会員 田原裕介*², 正会員 勝井辰博*², 川崎正人*³ 児玉欣二*³, 正会員 姫野洋司*²

Development of Large-Scale High-Performance CFD Coding Method for PC-Cluster Parallel Computing Environment

- 1st Report: Setup and Initial Evaluation of Coding Environment with MPI Protocol -

By Yusuke TAHARA (Member), Tokihiro KATSUI (Member), Masato KAWASAKI, Kinji KODAMA and Yoji HIMENO (Member)

This paper concerns development of large-scale high-performance (LSHP) CFD coding method for PC-Cluster Parallel Computing Environment. Main objective of the present study is enhancement of capability of Reynolds-averaged Navier-Stokes (RaNS) equation solver and its advanced applications for LSHP computing in ship hull form design. One of the finial goals is to achieve the scalable coding of RaNS equation solver in the level of 10⁷ computational grids using 64 CPUs. In the present 1st report, focus of discussion is placed on setup and initial evaluation of the present coding environment. The Message Passing Interface (MPI) protocol and the Score-D job scheduling method are implemented in the present system. In the following, details of the present parallel coding environment are described. Then results of the performance evaluation are discussed based on benchmark data for high-performance parallel coding of CFD-based nonlinear programming and a domaindecomposition-based commercial RaNS equation solver.

Keywords :

64-CPU PC Cluster, Parallel Computing, Large-Scale High-Performance Computing, MPI Protocol, RaNS Equation Solver, CFD-Based Nonlinear Programming

1. 緒言

最近の計算流体力学(CFD)の進歩は著しく,船舶海 洋工学の分野においても様々な応用や展開が試みられ ている.特にレイノルズ平均ナビエストークス方程式ソ ルバー(RaNS法)を用いた船体粘性流場の計算手法の開 発は活発であり,近年では自由表面影響を考慮できる RaNS法が通常のものとなりつつある.国内および海外 において,これらのRaNS法関連の研究を推進する背景 には、シミュレーション・ベースド・デザインの達成と いう目的がある.従来と全く異なるコンセプトの船型を 考える際,時間的および経済的制約がかつてと大きく異 なる現状においては,従来と同様な方法でモデルテスト による性能データベースを構築することは極めて困難 である.その打開策としてのシミュレーション・ベース ド・デザインを達成する一つの要素技術として,RaNS 法は極めて有望であると目されている¹⁾.

一方,近年のRaNS法利用の世界的動向は,より複雑 な流場を高精度に解析するために格子数 10⁶~10⁷レベ ルの中・大規模計算へ移行しつつあるにも関わらず,大 型メインフレームを利用できる特定の船型設計者以外 にとっては,かようなRaNS法の適用は極めて困難であ るのが実状である.また計算の大規模化は格子数の増大 のみに留まらず,例えば研究レベルでは既に達成されつ つあるRaNS法のより高度な利用法,一例として非線形

-360-

^{*1} 平成15年11月13日造船三学会秋季連合大会において講 演,原稿受付平成15年11月28日

^{*2} 大阪府立大学大学院

^{*3} 株式会社ケイ・ジー・ティー

計画法との結合による船型自動最適化などの大規模計 算手法を実際の設計現場で適用しようとする際にも,多 大な計算時間は重大なボトルネックとなっている.

上述したシミュレーション・ベースド・デザインの達 成に関する設計現場からの重要なニーズの一つは、比較 的安価な設備投資(数百万円程度)による大規模・高速計 算環境の実現である.その観点において,最近市場に供 給され始めたPCクラスターによる並列計算環境は極め て有望なソリューションであり、その戦略的基礎技術の 開発は国外2 だけでなく, わが国政府主導のプロジェク ト3)においても検討され、それらの成果はオープンソー スという形で公開されている.しかしながら,その環境 を十分に活用できるアプリケーション・コードは不足し ている傾向があると言われており4,5), これは船型設計 目的のRaNS法に関しても例外ではない¹⁾.上述したPC クラスターをはじめとするローエンド・ハイパフォーマ ンスな計算環境における,大規模問題の解法を視野に入 れた高効率RaNS法のコーディング技術の開発は、次世 代のCFD技術をIT (Information Technology)の進化に 適合させ,より高度なものに発展させていく意味におい て重要な課題である.

本研究の目的は,今後の応用拡大が予想されるPCクラ スター並列計算環境における大規模・高効率CFD,特に RaNS法とその高度利用法のコーディング技術の開発で ある. 以前盛んに行われた計算コードのベクトル化の場 合と異なり、並列計算の高効率化はコンパイラーオプシ ョンの発達だけでは達成できない^{5),6)}.本研究では、上述 した船型設計現場からのニーズを重視し,RaNS法とそ の高度利用法,特に複雑形状に対応するためのマルチブ ロックRaNS法(著者らの最近の事例として文献⁷⁾や RaNS法援用船型自動最適化^{®・10)}を視野に入れ、中・大規 模問題の高速解法に最も適合した並列計算環境の構築 とコーディング技術の開発を目指す.計算環境の構築に おいては,個々のハードウエア資源およびソフトウエア 資源のベンダー特殊性を極小化する目的で、基本的にオ ープンソースとして一般に公開されている技術を統合 するという方法を用いた. 最終的には 64CPU計算環境 においてもスケーラブル(scalable)であり、LINUX環境 のメッセージ・パッシング・インターフェース(MPI) 5).6) プロトコル対応の高効率コーディング技術の開発を目 標としている. このレベルのCPU数はハイエンド・モデ ルにも相当するが,数年後の計算資源のコストパフォー マンスの向上を予測し、かつそれに準備するという観点 では,一般デザイン部門への技術転用という目的でも妥 当な目標値であると判断している.

本研究の第一報となる本論文では、特に MPI プロト

コルを適用したコード開発環境の構築とその初期評価 について議論する.まず"粒度(granularity:計算時間と 通信時間の比)"という視点に基づき,連立非線形楕円型 偏微分方程式系の解法としての RaNS 法,さらに CFD 援用船型自動最適化手法の並列高速化に最も適したシ ステム構成を設計した結果について述べる.結論的には, メッセージパッシング方式が最も有望であると判断し, 実際にそのアーキテクチャーを実現する PC クラスター 並列計算環境を構築した.以下その詳細を述べるととも に,実際に非線形最適化問題の並列化コーディングを行 った結果や,汎用流場解析コマーシャルコードによるベ ンチマーク・テストの結果,ならびに本研究で改めて定 義する並列化率のモデル式に基づく計算パフォーマン スの予測法などを検討し,本研究の将来展開への準備を 行った結果について議論する.

2.システム構成の概要

2.1 並列計算の有効性

本研究で対象とするような大規模・高速計算(一般にス ーパーコンピューティングと総称)の達成度を直接的に 示す指標としては、従来から速度向上率が用いられてい る.例えばベクトル計算をスカラー計算と比較する速度 向上率 は、以下の式で定義されている.

$$R_{s} = \frac{1}{1 - \alpha + \frac{\alpha}{V}} \tag{1}$$

ここでαはベクトル化率, V(一般的に 10~20)はベクト ル演算性能向上率であり,後者はハードウエアとオペレ ーティングシステム・コンパイラーなどの性能によって ほぼ決定される値である.一方,並列計算の達成度も類 似した式で評価される.並列計算を単一 CPU の計算と 比較する速度向上率 R_s は以下の式で定義され,これは一 般にアムダールの法則と呼ばれている.

$$R_s = \frac{1}{1 - P + \frac{P}{n}} \tag{2}$$

ここでPは一個の CPU が実行する計算全体の中で並列 可能な部分, 1-Pは並列不可能な部分, そしてnは CPU 数である.本研究では、特にこのPを並列化率と定義す る.一般に、ベクトル計算,並列計算ともに問題の規模 が増大するに従って、計算全体に占めるベクトル化可能 部分や並列化可能部分の割合が高くなり、 α やPは増加 することが知られている.

ここで(1)式および(2)式の特徴的な傾向を見出すため に、両者に共通な式形である以下の関数の分布を Fig.1 に示す.

$$\phi = \frac{1}{1 - x + \frac{x}{x}} \tag{3}$$

まずベクトル計算を行った場合,現実的に最も高いベクトル演算性能向上率(ν =20)を仮定した場合でも,10倍の速度向上率 R_s を得るためには, α は 95%を越える必要がある.また仮に 100%の α が達成できたとしても,速度向上率は 20が上限となる.一方,並列計算の場合には,たとえ CPU数が無限大でも,並列化可能部分の割合が計算全体の 95%である場合(P=0.95,n= ∞), R_s は 20 で飽和する.しかしながらPを 100%に近づけるに従い R_s は CPU 数に対する線形増加に漸近し,CPU 数にほぼ比例した速度向上が図れることになる.問題の大規模化という近年の動向のもと,次世代のスーパーコンピューティングの技術開発は,並列計算ベースで行うことが不可避であるとされている.



Fig.1 Performance acceleration ratio of vector and parallel computations.

一方,問題の規模(例えば格子数)をm,単一 CPU に よる計算時間を $T_s(m)$, CPU 数nによる並列計算を行っ た場合の計算時間を $T_p(m,n)$ としたとき,並列計算の効 率E(m,n)は以下のように定義されている.

$$E(m,n) = \frac{R_s(m,n)}{n} = \frac{T_s(m)}{nT_P(m,n)}$$
(4)

ある並列計算コードが任意のnに対して *E(m,n)* を一定 と出来るようなmを見出せるとき, このコードはスケー ラブル(Scalable)であるという.本研究で目的とするの はこのような RaNS 法およびその高度利用法のコーデ ィング技術の開発である.



(b) The distributed memory (message passing) model. Fig.2 Comparison of parallel computing models.

2.2 並列計算モデルの検討: MPI プロトコルの採用

高効率の並列化計算を行うためには、解くべき問題と 利用環境に最も適合した並列計算環境を設計する必要 がある.ここでは代表的な並列計算モデルを検討し、最 終的に本研究で選択するモデルを決定した過程を述べ る.現行の並列計算モデルは、大別してFig.2に示す(a) 共有メモリ型モデル(Shared memory model)および(b) 分散メモリ型モデル(Distributed memory model)に分 類でき、その主な特徴は以下のように要約することがで きる.

 (a) 共有メモリ型モデル:複数のプロセス(CPU)が単一 のアドレススペース(メモリースペース)を共有す るモデルであり、ハイエンドの特に大型コンピュー タシステムで採用されている. CRAY-X-MP システ ムは最初の共有メモリ型パラレル・ベクトル計算機 であるとされている. CFD コーディングのツールと しては、この方式に対応した FORTRAN の拡張とし て OPEN MP があり、FORTRAN からの API (Application Program Interface)として提供されて いる.

(b) 分散メモリ型モデル:複数のプロセスが個々のロー カル・アドレススペースを有するモデルであり、そ れらは通常高速ネットワークで結合されている.メ ッセージパッシング(Message-passing)方式とも呼 ばれる. 近年 PC (Personal Computer)や EWS (Engineering Workstation)の高機能化に伴い,より 多く採用されはじめたモデルである. この方式に対 応した CFD コーディングのツールとしては, FORTRAN の拡張として HPF(High Performance Fortran)や MPI (Message-passing Interface)がある. 前者はデータをどのように分散メモリ上に配置する かを指定し、コンパイラがデータ構造に従い、並列 化可能な DO ループを自動的に並列化する.一方後 者は FORTRAN コーディングの段階より Master/Slave 形式で並列プロセスを制御すため、よ り効率的に並列化を達成する. この MPI 方式の利点 の詳細については後述したい.

ここで本研究の目的であるデザイン部門でも容易に 適用できる並列計算環境の構築という観点で検討する と、明らかに分散メモリ型モデルが有利であることが分 かる.上述したとおり、分散メモリ型モデルは共有メモ リ型モデルに比べはるかに安価に構成でき、またハイエ ンドシステム特有の大規模な設備投資の必要がない.ま た近年の PC や EWS の性能向上に伴い、事実上個々の プロセスが利用できるキャッシュやメインメモリの規 模が共有メモリ型モデルと同等か、場合によってはより 大きくなっている.さらにこのモデルにおいては個々に 性能のばらつきがある CPU をネットワークに接続して 利用することができ、結果的にこの特徴は、比較的小規 模の追加設備投資額によって、必要に応じてシステムを 拡張するという方法を可能にするものである.

一方,並列計算において粒度(granularity)の概念は極 めて重要である.各CPU間のロードバランスが保たれて いると仮定した場合,粒度は個々のCPUの計算時間と通 信時間の比(計算時間/通信時間)で定義され,粒度が小さ い細粒度(fine grain)の場合は個々のCPUでは計算に比 較して通信の負荷の割合が大きくなるのに対し,大粒度 (coarse grain)の問題ではその通信負荷は無視できる.従 って,特に細粒度の計算の並列処理で効果を上げるため には高速なCPU間通信が必要となるが,大粒度の場合の 通信速度はさほど問題にならない.共有メモリ型モデル は主記憶を通じてデータ転送が行われるためデータ転 送速度は極めて高く、細粒度の問題の並列化に適する. 一方、分散メモリ型モデルは一般にデータ転送速度が共 有メモリ型より低いため、中粒度・大粒度の問題の並列 化に適している.本研究で対象とするRaNS法は、その 基本的な特徴として連立非線形楕円型偏微分方程式系 の解法に基づいており、領域分割法の導入と問題規模の ある程度の上方維持により、容易に中粒度もしくは大粒 度の問題に転換できる¹¹⁾.加えて一般的なCFD応用船型 自動最適化の並列化を考えた場合、最も計算時間を消費 する目的関数の計算部分を並列処理するアルゴリズム を用いれば、この問題を極めて粒度の大きい並列化問題 に転換できる(詳細は後述).このような理由においても、 本研究の目的に合致した並列計算モデルは分散メモリ 型モデルであると結論できる.

次に、分散メモリ型モデルで使用する並列処理のプロトコルとして、MPIの利点に着目する.既述したとおり、 HPFに比べMPIはより効率的な方法であり、その特徴についてはGroppら[®]が詳しい. GroppらはMPIの利点および特徴として以下の項目を挙げている.

- (A)表現度(Expressivity): MPI 方式は他の方式に比べ, より完全な形で並列処理を既述できる. HPF などの データ構造に従う並列化や,コンパイラー依存型の 並列化で不足していた機能がより明確な形で定義で きる.
- (B) コード修正度(Easy of debugging):並列化コーディングにおいてデバッグは最も労力を費やす作業のひとつであり、共有メモリ型対応 OPEN MP などで頻発するメモリ重複に関するエラーから完全に解放される.これはコードの生産性を著しく向上させる重要な要素となりえる.
- (C) パフォーマンス(Performance): 既述したとおり, 分散メモリ型モデルに準じる MPI 方式の弱点は通信 速度の限界であるが,近年個々のプロセッサーやネ ットワーク技術の高性能化および低価格化が進んだ ことによって,これらの問題が低減されつつある. 既述した粒度の問題にさえ注意すれば,MPI 方式に おいても十分なパフォーマンスが期待できる.

以上の検討の結果、本研究では分散メモリ型モデル対応 MPI 方式を採用することに決定した. なお実際のシステム構成を設計した際、概念的には分散メモリ型モデルの拡張であり、構成的には共有メモリ・分散メモリ複合型モデル(Combined model)と定義されるべき Fig.3 に示すモデルを用いることにした. このモデルは上述し

た2つのモデルの特徴を合わせもつものであり,単純な 分散メモリ型モデルよりもコスト・パフォーマンスの観 点で利点がある.それは個々のプロセス(もしくは CPU) をその支援的ハードウエア資源(キャッシュメモリや磁 気ディスク,冷却機構など)とともに物理的に全て独立さ せるより,共有できる部分をまとめる方がより効率的で あり,事実上個別のパーツ数を減少させることによる故 障頻度の低下も期待できる.このFig.3に示すモデルを 用い,オペレーティング・システムも含むソフトウエア 構成の設計は分散メモリ型モデルとして行った.その詳 細については次節で述べる.



Fig.3 The present combined model for parallel computing.

2.3 本研究における並列計算環境の構成

ここでは本研究において構築した並列計算環境の概 要を述べる.その基本的な構築方針は前節で述べたとお りである.まず Fig.4 にネットワーク構成の概観を示す. Fig.4-a は初期導入の段階であり,当初は 24CPU 構成の PC クラスター並列計算環境を構築した.現在は Fig.4-b に示す 64CPU 構成への移行を達成しつつあり,本稿執 筆段階では 44CPU 構成の検証までが完成している.

Fig.5 および Fig.6 にはそれぞれ並列計算ジョブ・スケ ジューリング法およびオペレーティング・システム(OS) などのソフトウエア・レイヤー構造を示す.特に今回の ジョブ・スケジューリング法として特筆すべき要素は SCore-Dの採用であり,他の手法と比較したその特徴に ついては後述する.なお既述したとおり,これらのソフ トウエアの重要な部分は全てオープンソース・レベル(我 が国経済産業省(当時通産省)主導のプロジェクトがその 後継承された PC Cluster Consortium が管理・提供)の ものであり基本的に LINUX 上で稼動する.以下でその 主要ソフトウエアの概略を述べる.

○ PM II (PMv2): クラスタ型並列計算用の高速通信通 信ライブラリーである. このライブラリーは、クラ スタコンピューティングのために設計開発され, 10/100/1000 Ethernet および Myrinet ネットワーク 上で稼動する. 共有メモリ機構を用いて SMP (Symmetric Multi Processing)上の通信にも PMv2 を使用することができる. また PMv2 の上に構築し た MPI 通信ライブラリ(MPICH-SCore)が提供され ている.

- SCASH: PM II を用いたソフトウエア DSM
 (Distributed-Shared Memory)システムである.
- MPICH-SCore: ワークステーションや PC, さらに SMP クラスタで稼動する MPI であり、その環境に おいては現在最も高性能であると考えられている。
- PGI コンパイラ:米国 ASCI(Accelerated Strategic Computing Initiative)プロジェクトで採用された Intel アーキテクチャプロセッサ用に最適化されたコンパイラである.
- SCore-D: プロセッサやネットワークのような, クラ スタのリソースを利用するユーザレベルのグローバ ル・オペレーティング・システムである. 種々のク ラスタ型並列計算用・高速通信通信ライブラリー対応のネットワークデバイスを直接的・統一的に管理 する.

ここで特にジョブ管理の方法に着目し、今回採用を決 定した SCore-D システムの特徴を述べる. Fig.5 におい τ , (a) SMP (Symmetric Multi Processing), (b) Beowulf Cluster, そして今回使用した(c) SCore システ ムを比較している.まず SMP (Fig.5-a)は共有メモリー 型モデルで使用されてきた手法であり、それに対して Beowulf Cluster (Fig.5-b) は NASA (米国航空宇宙局) の Beowulf プロジェクトで確立された分散メモリー型 モデル用の方式である. Beowulf Cluster はシステムソ フトウエアとしては存在せず,汎用 LINUX の諸機能を 利用して並列計算環境を構築する.従来はこの手法が多 く用いられていたが、管理の煩雑さなどの問題が指摘さ れてきたことを背景に SCore (Fig.5-c)が開発された. SCore は PC Cluster Consortium において Beowulf Cluster の難点を改良する目的で開発され、システム運 営管理負担の低減,固有プロトコルによる通信速度の向 上などの利点が認められている.

以上のシステム構成で PC クラスター並列計算環境を 構築し,本研究の主目的である並列化コーディング技術 の開発を行う準備を完了した.ハードウエアを含むシス テム構成の概要を Fig.7 に示す.この計算環境の初期評 価を行った結果については次章で議論する.



(b) Final installation Fig.4 The present PC-cluster network configuration.

-365-



(a) SMP system (b) Beowulf cluster system (c) SCore system Fig.5 Comparison of Job-scheduling methods.



Fig.6 Software layer structure of the present PC-cluster system.



Fig.7 Overall configuration of the present PC-cluster system.

-366-





Fig.8 Comparison of serial and parallel computation architectures for SQP algorithm: rows, serial architecture, and parallel architecture, respectively.



Fig.9 Parallel-Based coding for the present SQP algorithm.

3. 並列計算環境の初期評価

3.1 非線形計画法の並列化コーディング

ここでは、前章で論じた本研究の並列計算環境におい て、実際に非線形最適化問題の並列化コーディングを行 った結果を検討し、本環境の初期評価を試みる.対象と したのは逐次二次計画法(SQP)に基づく非線形最適化問 題である. Fig.8 および Fig.9 には、それぞれ並列化の アーキテクチャーと通常の SQP アルゴリズムにおける 並列化部分を示す. CFD 手法を SQP とともに用いる場 合,最も計算時間が消費されるのは目的関数の評価であ り,その個々の評価において CFD が実行されると考え てよい.一般的な SQP アルゴリズムにおいては,一回 の最適化サイクルが完了するまでには,線形探索(1-D Search)および近似ヘッセ行列・最適点目的関数の更新 (Update f and Approximate Hessian Matrix B)とい う2個所で一連の目的関数の評価が実行される.通常両 過程とも複数の設計変数に対する目的関数を計算して おり,特に後者においては目的関数の気配を求める中心 差分オペレーションと最適点関数の更新によって総計 2k+1回(kは設計変数の総数)だけ CFD が実行される.



Fig.10 Parallel-Based Optimization test case 1: layout optimization problem of 2-D tandem hydrofoils under free surface (minimization of wave-making resistance with minimum lift constraint).

今回の並列化コーディングにおいては、MPI プロトコ ルを用い、Master プロセス(Proc.0)に SQP アルゴリズ ムの制御を、そして Slave プロセス(Proc.1~m)に関数 評価を行うように並列化することにより、この部分にお ける計算速度の向上を目指した. MPI プロトコルに基づ くコーディングでは、Master プロセスと Slave プロセ スに同一のプログラムが利用でき、計算を実行する際は 全てのプロセスを同時に開始する.その後各プロセスは 個々に認識した自分のプロセス ID(0~m)に従い、ID が 0 の場合は Master プロセス用の、そして ID が 1~m の 場合は Slave プロセス用の作業が記述されたサブルーチ ンをコールし,各々を実行する.なお各プロセスは独立 した CPU を占有するように設定する.

全てのプロセスには CFD 計算に必要な物体幾何情報 や流場情報を並列的に保有させるため、プロセス間通信 の情報としては CFD 実行開始命令・終了通知、設計変 数値、そして目的関数値・制約条件関数値のみに限定さ れ、全体の計算時間に占めるネットワーク通信の時間は 無視できるほど小さい.また今回の例では、1 最適化サ イクルにつき Master プロセスが管理する SQP実行時間 は Slave プロセスが管理する CFD 実行時間にくらべ4 オーダー以上小さい.従ってこの並列化では、目的関数 評価以外の部分の計算時間は極めて小さく、また特に Slave プロセス間ではロードバランスが均一に保たれ、 既述した粒度の概念においては大粒度の問題の一種で あると定義できる.



Fig.11 Parallel-Based Optimization test case 2: optimization of stern frame lines of tanker hull form (minimization of viscous resistance with displacement and engine space constraints).

この並列化ベースのSQPモジュールを用い,以前著者 らが行った2つの非線形最適化問題,すなわち自由表面 下の複翼最適配置問題(最終結果Fig.10)¹²⁾と,粘性抵抗 最小化を目指したタンカー船型の船尾形状最適化問題 (最終結果Fig.11)⁸⁰を再計算した.計算条件の詳細につい てはそれぞれの文献を参照されたい.両問題とも設計変 数の総数は6であり,使用したCPU数はMasterプロセ ス用の1個と目的関数・制約条件関数評価プロセス用に 13(=2*k*+1)個の総数14個である.検討の結果,計算結果 が同一であることはもとより,当初期待したとおりの速 度向上値,すなわち約13倍弱の計算速度向上が達成で きていることを確認した.完全に2*k*+1倍にならない理 由は,小規模ながらもI/Oや通信負荷の影響が存在する ためであると考えている.この結果より,本研究で実践 した並列計算がCFD援用非線形計画法の高速化に極め て有効であることを再確認できたとともに、この応用事 例における本計算環境の有効性が確認できたものと考 える.



Fig.12 Computational grid around three-dimensional wing geometry.

3.2 汎用流場解析コードを用いた並列計算環境の評価

本研究では、将来的には格子数 10⁶~10⁷レベルの中・ 大規模問題においてもスケーラブルなRaNS法を実現す るためのコーディング技術の開発を目的としている.そ れを実践するRaNS法カーネルには、近年著者らが開発 し、その有効性を確認したマルチブロック計算格子対応 RaNS法⁷⁾を用いる予定である.ここではその事前準備と して、類似した機能を有するコマーシャルコード、 FLUENT Ver.6.1.22¹³⁾を用いたベンチマーク・テストの 結果を検討する.

本計算コードは、与えられた利用可能CPU数に応じて ロードバランスの均一性を考慮しながら自動的に計算 領域を分割し、並列計算を行う機能を有する.本検討の ために, NACA 0012 翼型まわりの2次元計算格子をス パン方向に拡張した7万点および70万点の三次元格子 (以下それぞれ 70K MESHおよび 700K MESH)を準備 した.本研究においては、それらは小規模問題および中 規模問題に位置付けられている. 70K MESHの概観を Fig.12 に示す. 目的はあくまでパフォーマンス評価であ るので、比較的安定した計算が可能な流場条件を設定し ている. 設定レイノルズ数は 10⁶, 迎角は 0°, 乱流モ デルには k-εモデル,運動量方程式対流項の離散化には QUICKスキーム, $k-\varepsilon$ モデルの両方程式対流項の離散 化には2次精度風上差分スキーム,速度場・圧力場結合 にはSIMPLE法を用いている. なお以下で論じる実行時 間の計測においては、各CPU数において同一初期流場よ り100時間ステップの計算を10回行って実行時間を計 測し、その平均値をそのCPU数における計測値として用 いている.

上記の条件において, CPU 数 *n* =1~40 で実測した速 度向上率 *R_s*, そしてその *R_s*を用い(2)式より逆算した並 列化率 *P*, すなわち

$$P = \frac{1 - 1/R_S}{1 - 1/n} \tag{5}$$

さらに本研究で検討したモデル関数による分布を Fig. 13 に示す.検討した結果,両格子について以下のモデル 式を用いる方法が実測値の傾向を最も良く表現できる ことがわかった.

$$\begin{cases} P = -a_1 \ln(n) + a_2 \\ R_S = \frac{1}{1 - P + \frac{P}{n}} \end{cases}$$
(6)

ここで*a*1 および*a*2 は正の定数である. *P* が対数的に減 少する理由としては、並列計算におけるプロセス間通信 がツリー構造的に行われている場合、その通信量がプロ セスの増加にともなって対数的に増加することに関連 していることが考えられる. このモデル関数の検討は次 節でも行う.



Fig.13 Measured parallel-computing performance for FLUENT (Ver.6.1.22) code.

Fig. 13 に示されているように,両格子の結果はとも に n の増加に対し R_sの勾配が減少する傾向を示すが,これは問題規模を限定して n を増加した場合に通常発生

する現象として認識されているものである. nの増加に 対する R_sの勾配の減少は, nの増加とともに領域分割数 が増加することによって,個々のプロセスが扱う問題規 模が縮小するとともにその計算時間が減少するのに対 し,領域境界面の数の増加によって境界間通信の時間が 増加する,すなわち粒度の縮小による並列計算効率の低 下によるものとも捉えることができる.

一方,70K MESH と700K MESH の結果が示す明ら かな傾向の違いは、本計算環境を評価する上で極めて重 要なものである.70K MESH の R_s は比較的早い段階で 減少する傾向が見られるのに対し、700K MESH の R_s は 常に増加傾向にある.本計算環境の全資源を同時活用す る観点で考えると、この結果は、本システムは700K MESH レベルの中規模問題において有効であり、さらに 規模の大きい問題においてはより有効な速度向上率を 得ることができる可能性を示唆しているように思われ る.これは本並列計算環境の設計段階で目指したものと 合致する傾向であり、本システムが所期の目的どおりの 特性を有することを裏付けるものであると考えている.

なお,ここで行ったコマーシャルコードによるベンチ マーク・テストの結果は,本研究で将来的に行う高効率 並列化コーディングにおいて,妥当な目標値・比較参考 値として活用する予定である.

3.3 実用的な並列計算パフォーマンス評価法およびパフォ ーマンスのモデル化の検討

ここでは、本研究で改めて定義した並列化率 P と速度 向上率 R_sを用い、実用的な観点に基づく並列計算パフォ ーマンスの評価法およびパフォーマンスのモデル化を 検討した結果について述べる. 簡便のため、以下の議論 においては全ての変数を実数として扱う. まず計算効率 に関する検討を行うために、(2)式と並列計算効率 E(m,n) を定義する(4)式より以下の関係式を得る.

$$E(m,n) = \frac{R_S(m,n)}{n} = \frac{1}{n(1-P) + P}$$
(7)

ここで通常 0 < P < 1 である. すなわち, P が一定であっ ても E(m,n) は n の増加とともに減少することになる. 既 述したとおり,スケーラブルなコードとは n の増加にお いて E(m,n) の低下をもたらさない問題規模 m を決定で きるコードである.以下,スケーラブルであるための条 件をさらに検討するために $n = n_0$ および $m = m_0$ において E(m,n) の低下がない条件より以下の関係を考える.

$$\delta E(m,n) = \frac{\partial E(m,n_0)}{\partial m} \delta m + \frac{\partial E(m_0,n)}{\partial n} \delta n = 0$$
(8)

さらに(8)式と(7)式の E(m,n) の定義式より以下の関係式 を得る.

$$\frac{\partial R_{\mathcal{S}}(m,n_0)}{\partial m}\bigg|_{m=m_0} = \left(\frac{R_{\mathcal{S}}(m_0,n_0)}{n_0} - \frac{\partial R_{\mathcal{S}}(m_0,n)}{\partial n}\right)\frac{\delta n}{\delta m}$$
(9)

上式は、スケーラブル・コーディングにおいて速度向上 率 R_s が満たすべき条件となる. ここで R_s の n に関する 線形性が仮定できる場合、すなわち $\partial R_s(m_0,n)/\partial n = \beta_0$ で あるような定数 β_0 と $R_s(m_0,n_0) \approx 1 + (n_0 - 1)\beta_0$ が仮定でき るとすれば、より簡便な以下の式が導かれる.

$$\frac{\partial R_s(m,n_0)}{\partial m}\bigg|_{m=m_0} = \frac{1}{n_0} (1-\beta_0) \frac{\delta n}{\delta m}$$
(10)

すなわち,固定したnについて R_s のmに関する勾配を評価すれば,スケーラブル・コーディングを実証するために拡大すべきmの増分 *om*を推定することができる.上式はスケーラブル・コーディングを評価する際の有用な関係式になり得ると考えられる.



Fig.14 Ideal and measured parallel-computing performance curves.

次に与えられた並列計算環境・問題規模・計算アルゴ リズムにおいて、期待できる速度向上率と並列化率の上 限を検討する. Fig.14 には、前節で検討した 700K MESH の $R_s \ge P$, さらに区間 n = [1,2]で評価した R_s の勾 配を維持し、n に関して線形的に増加する理想的な Ideal $R_s \ge$ Ideal Pを示す. Ideal $R_s \ge$ Ideal Pは以下のよ うに導くことができる.

$$Ideal \begin{cases} P = \frac{n}{\frac{1}{\beta_{1}} + n - 1} &, \quad \beta_{1} = R_{S}|_{n=2} - R_{S}|_{n=1} < 1 \\ R_{S} = \frac{1}{1 - P + \frac{P}{n}} \end{cases}$$
(11)

ここで β, は n = [1,2] で評価した R_s の勾配であり,実測に 基づいて決定できる. β, は並列計算環境・問題規模・計 算アルゴリズムに依存すると考えられ,実用上最も重要 な特性値である.問題規模と計算アルゴリズムが固定さ れた場合は, β, は並列計算環境のバックボーン LAN の 速度が向上すれば増加し,CPU 単体のクロック数など の性能が向上すれば減少する. Fig. 14に示す Ideal R_s は、 与えられた並列計算環境・問題規模・計算アルゴリズム において達成できる R_s の上限を示し、また Ideal Pは それに対応する並列化率を示す. 各nについて Ideal R_s および Ideal Pに対する実際の R_s およびPの比率を評 価すれば、各nにおける並列計算高速化の達成度を客観 的に示すことができると考えられる.

最後に並列計算パフォーマンスのモデル化について 述べる.前節の検討に基づき,固定した問題規模 m₀に おける並列化率 P が対数式を用いてモデル化できると 仮定し,改めて以下の式を定義する.

$$P(m_0, n) = -\alpha \ln(n) + \beta$$

$$R_S(m_0, n) = \frac{1}{1 - P(m_0, n) + \frac{P(m_0, n)}{n}}$$
(12)

ここでモデル定数 α (>0) と β (>0), R_s , そして P には以下のような関係を見出すことができる.

$$-\alpha = \frac{\partial P}{\partial n}\Big|_{n=1} < 0 \quad , \quad \beta = P\Big|_{n=1} = \frac{\partial R_S}{\partial n}\Big|_{n=1} < 1 \tag{13}$$

このモデル定数を決定するためには、 $2 \le n$ の異なる 2ケース、すなわち n_1 および n_2 のベンチマーク・テストを 行えばよく、その結果得られた R_s より(5)式を用いて P_1 および P_2 を求める、結局、 $\alpha \ge \beta$ は以下のように簡単 に決定することができる.

$$\begin{cases} \alpha = \frac{P_1 - P_2}{\ln n_2 - \ln n_1} \\ \beta = \frac{P_1 \ln n_2 - P_2 \ln n_1}{\ln n_2 - \ln n_1} \end{cases}$$
(14)

ここで(13)式にも示すように、Pon=1における値は R_s の初期勾配となり、これはPの式形に依存しない関係である.上記のモデル式を用いれば、それが β で与えられる.本モデル式においては、 β および α はそれぞれ R_s の初期および中・後期の挙動を特徴づけるパラメータであると言え、これらのパラメータを決定する要素には計算環境、問題規模、そしてコードのアルゴリズムが挙げられる.それらとモデル定数の相関に加え、本モデル化の方法の妥当性については今後さらに検討していく.これらのモデル化が妥当なレベルで成功すれば、例えば設計部門における現行システムの拡張を検討する場合の費用対効果を明確にすることができ、極めて有用である.

なお、ここで行ったような並列計算パフォーマンスの モデル化は、比較的簡単な問題や、特定のアルゴリズム に基づく数値解法に関しては従来検討されてきたが、本 研究の主眼となる 3 次元 RaNS 法(領域分割型・陰解法 ベース PISO アルゴリズム等)を対象とした詳細な研究 事例はこれまでに報告されておらず、今後さらに検討を 進めるべき重要な課題であると考えている.

4. 結営

本研究の目的は, PC クラスター並列計算環境におけ る大規模・高効率 CFD とその高度利用法のコーディン グ技術の開発であり、本研究の第一報となる本論文では、 特に MPI プロトコルを適用したコード開発環境の構築 とその初期評価について議論した.本研究の目的に適し た環境を設計した結果、メッセージパッシング方式が最 も有望であると判断し、実際にそのアーキテクチャーを 実現する PC クラスター並列計算環境を構築した.

本研究で構築した並列計算環境において実際に非線 形最適化問題の並列化コーディングを行った結果,本並 列的手法の CFD 援用非線形計画法の高速化に関する有 効性と同時に,本並列計算環境の有効性が確認できた. また汎用流場解析コードによるベンチマーク・テストを 行い,将来参照可能な目標値を得ることに加え,本計算 環境の全資源の同時活用という観点では本システムは 中規模問題について有効であり,より大規模な問題にお いてはさらに有望であるという認識を得た.最後に本研 究独自の並列計算の評価法とパフォーマンス・モデルを 提案し,その基礎的検討までを完了した.

結論として,これまでの評価結果に基づく限り本研究 で構築した並列計算環境は所期の目的どおりの性能を 有し,今後実際に検討していく大規模・高効率RaNS法 とその高度利用法のコーディング技術の開発を行う基 礎段階が完了したものと考える.なお冒頭に述べたよう に,本研究の重要な意義の一つは次世代のCFD技術をIT の進化に適合させ,より高度なものに発展させていくこ とである.同様な目的に類する他分野の成果(構造解析に 関する最近の事例として文献¹⁴⁰など)も十分に検討しな がら,今後研究を進展させていく予定である.

謝辞 辞

本研究を行うにあたり種々のご助言ご協力をいただ きました大阪府立大学大学院工学研究科・助教授・正岡 孝治先生,ならびにFLUENT Ver.6.1.22 を用いたベン チマーク・テストにおいてご協力を頂きましたユニバー サル造船株式会社技術研究所・牧野功治氏,フルーエン ト・アジアパシフィック株式会社・桑山智一氏に感謝の 意を表します.なお本研究の一部には文部科学省研究費 補助金(2002・2005, Project Number 14350524; 2001・2003, Project Number 13555272),および The U.S. Office of Naval Research (2002・2005) Award Number N0014-02・1・0256)の援助を受けたことを記し, 関係各位に感謝いたします.

参考文献

- Proceedings of Workshop on Numerical Ship Hydrodynamics (Gothenburg 2000), Gothenburg, Sweden, 2000.
- 2) Accelerated Strategic Computing Initiative, http://www.tasc.com/tnm/messina/sld005.htm.
- PC Cluster Consortium, http://pdswww.rwcp.or. jp.
- 三好俊郎,坂田信二,吉田有一郎,斉藤直人:スーパー コンピューティング, 培風館, 2001.
- Gropp, W., Lusk, E., Skjellum, A.: Using MPI -Portable Parallel Programming with the Message-Passing Interface, The MIT Press, 1994.
- Pacheco, P.S.: Parallel Programming with MPI, Morgan Kaufmann Publishers, 1997.
- 7) 田原裕介,林豪:マルチブロック NS/RaNS 法による アメリカ杯レース艇用風下帆走セールシステム周 りの流場解析,日本造船学会論文集,第 194 号, 2003.
- 8) 田原裕介,齋藤泰夫,姫野洋司:CFDによるタンカ ー船型の船尾形状最適化-第1報:粘性抵抗最小化 -,関西造船協会論文集,第231号,pp.29-36, 1999.
- 9) 田原裕介,齋藤泰夫,松山博志,姫野洋司:CFDによるタンカー船型の船尾形状最適化-第2報:伝達馬力最小化-,関西造船協会論文集,第232号,pp.9-18, 1999.
- 10) 田原裕介,西田隆司,安東潤,姫野洋司:CFDによる タンカー船型の船尾形状最適化-第3報:自航シミ ュレータを用いた伝達馬力最小化-,関西造船協会 論文集,第234号,pp.41-50, 2000.
- Smith, B., Bjorstad, P., Gropp, W.: Domain Decomposing – Parallel Multilevel Methods for Elliptic Partial Differential Equations, Cambridge University Press, 1996.
- 12) Kitamura, T., Tahara, Y., Himeno, Y.: A Study on Layout Optimization Problem of 2-D Tandem Hydrofoils under Free Surface, J. Kansai Society of Naval Architects, No. 228, pp. 67-78, 1997.
- 13) FLUENT 6.1 User's Guide, Fluent Inc., 2003.
- 14) 正岡孝治:インターネットソケット技術による矩形板の座屈応力評価法,関西造船協会論文集,第240号, pp. 159-164, 2003.

INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS Int. J. Numer. Meth. Fluids 2006; **52**:499–527 Published online 24 February 2006 in Wiley InterScience (www.interscience.wiley.com). DOI: 10.1002/fld.1178

CFD-based multi-objective optimization method for ship design

Yusuke Tahara*,[†], Satoshi Tohyama[‡] and Tokihiro Katsui[§]

Osaka Prefecture University, Japan

SUMMARY

This paper concerns development and demonstration of a computational fluid dynamics (CFD)-based multi-objective optimization method for ship design. Three main components of the method, i.e. computer-aided design (CAD), CFD, and optimizer modules are functionally independent and replaceable. The CAD used in the present study is NAPA system, which is one of the leading CAD systems in ship design. The CFD method is FLOWPACK version 2004d, a Reynolds-averaged Navier-Stokes (RaNS) solver developed by the present authors. The CFD method is implemented into a self-propulsion simulator, where the RaNS solver is coupled with a propeller-performance program. In addition, a maneuvering simulation model is developed and applied to predict ship maneuverability performance. Two nonlinear optimization algorithms are used in the present study, i.e. the successive quadratic programming and the multi-objective genetic algorithm, while the former is mainly used to verify the results from the latter. For demonstration of the present method, a multi-objective optimization problem is formulated where ship propulsion and maneuverability performances are considered. That is, the aim is to simultaneously minimize opposite hydrodynamic performances in design tradeoff. In the following, an overview of the present method is given, and results are presented and discussed for tanker stern optimization problem including detailed verification work on the present numerical schemes. Copyright © 2006 John Wiley & Sons, Ltd.

KEY WORDS: CAD; RaNS solver; hull-form optimization; multi-objective genetic algorithm; maneuverability; propulsion

1. INTRODUCTION

Currently, computational fluid dynamics (CFD) is used as an analysis tool to study alternative ship hull-form designs. Although extremely valuable, this approach suffers the limitation that

[†]E-mail: tahara@marine.osakafu-u.ac.jp

Contract/grant sponsor: U.S. Office of Naval Research; contract/grant numbers: N000140210256, N000140510616

Received 30 August 2005 Revised 12 December 2005 Accepted 14 December 2005

Copyright © 2006 John Wiley & Sons, Ltd.

^{*}Correspondence to: Y. Tahara, Marine System Engineering, Osaka Prefecture University, Sakai, Japan.

[‡]E-mail: Satoshi_Tohyama@marine.osakafu-u.ac.jp

[§]E-mail: kastui@marine.osakafu-y.ac.jo

it does not identify the optimum design. This is the background for developing CFD-based optimization methods wherein automatic determination of optimum shape is part of the simulation. Such approaches with complex CFD analysis have been developed by the present authors and colleagues [1-12], where the main emphasis is placed on utilization of advanced geometry modelling and high-fidelity Reynolds-averaged Navier–Stokes (RaNS) equation solver [1-10], including consideration of complexity of a real-life design problem, conditions, and constraints, and comprehensive evaluation of the results through model test verification [11, 12]. Besides works cited above, other recent applications of CFD-based optimization [13-18] witness that optimal shape design is receiving growing consideration in the naval hydrodynamics community, filling the gap with other fields (automotive, aeronautical, etc.) at a fast pace.

One of important components of the CFD-based optimization is a geometry modelling method to provide a link between the design variables and a hull form. More specifically, the parametric expression and modification of the ship hull form is an essential feature for the automatic optimization. However, in most of the related studies, solutions of the optimization schemes are not directly linked with computer-aided design (CAD) model actually used by hull-form designers (see Reference [5] for more complete survey). The limitation leads to a fact that modification trends proposed by the optimization schemes are often far from those designers can accept. More advanced and capable approach must be introduced in the optimization scheme, i.e. implementation of the direct link with designer's CAD system for the CAD-based hull-form modification is necessary.

On the other hand, the optimal design of the hull shape is basically a multi-criteria (or multiobjective) problem. For instance, goals of the design process can be resistance reduction, low noise, minimal wave height, reduced amplitude and acceleration of particular motions, etc. In addition, ship designers may also be interested to enhance certain quantities related to the engine power or to the maintenance costs. Unfortunately, the improvement of a specific aspect of the global design usually causes the worsening for some others, that is to say the optimal design of a ship hull is a multi-objective optimization problem. Therefore, the correct approach to the problem must follow the multi-objective optimization theory and the present work represents an attempt to develop such a procedure for ship design, involving the modelling, the development and the implementation of algorithms for the hydrodynamic optimization.

This paper concerns development and demonstration of a CFD-based multi-objective optimization method for ship design. Main objective of the present study is a system development and demonstration of the capability, which justifies use of simplified conditions, e.g. static sinkage and trim, and simplified design constraints. Three main components of the method, i.e. CAD, CFD, and optimizer modules are functionally independent and replaceable. The CAD used in the present study is NAPA system [19], which is one of leading CAD systems in ship design. In combination with the CAD model, a practical hull-form modification method is proposed, i.e. that is based on one-parameter hull-form blending and two-parameter prismatic curve control, where the latter is an extension of an original scheme used in may shipyards [20]. The CFD method is FLOWPACK version 2004d, a RaNS solver developed by the authors [21–23]. The CFD method is implemented into a self-propulsion simulator [23], where the RaNS solver is coupled with a propeller-performance program based on infinitely bladed propeller theory [24] in an interactive and iterative manner. In addition, a maneuvering simulation model is developed and applied to predict ship maneuverability performance, where the results are verified through comparison with experimental data [25]. Two nonlinear optimization algorithms are used in the present study, i.e. the successive

Copyright © 2006 John Wiley & Sons, Ltd.

CFD-BASED MULTI-OBJECTIVE OPTIMIZATION METHOD

quadratic programming (SQP) [1–5] and the multi-objective genetic algorithm (MOGA) [12, 26], while the former is mainly used to verify the results from the latter. For demonstration of the present method, a multi-objective optimization problem is formulated where ship propulsion and maneuverability performances are considered. That is, the aim is to simultaneously minimize opposite hydrodynamic performances in design tradeoff, i.e. delivered horsepower (DHP) and the first overshoot angle in (OSA) 10/10-degree-Zigzag test obtained from the self-propulsion simulator and maneuvering simulation method, respectively. In the following, an overview of the present method is given, and results are presented and discussed for tanker stern optimization problem including detailed verification work on the present numerical schemes.

2. OVERVIEW OF COMPUTATIONAL METHOD

To develop CFD-based optimization methods, three main components must be built and are common among many different applications (see Figure 1): first, a method to solve the nonlinear optimization problem formed by the objective and the constraint functions; second, a geometry modelling method to provide a link between the design variables and a body shape; and third, a CFD solver used as analysis tool to return the value of the objective function and of functional constraints. In the present study, a CAD-based hull-form modification method will be adopted. Two approaches are possible, i.e. CAD direct control and CAD emulation approaches. Those are illustrated in Figure 2, i.e. systems 1 and 2, respectively. In the former, optimizer directly executes CAD macro-file in which the procedures of hullform modification, geometry analysis, and CFD pre-processing are described. In the latter, a module is implemented in order to emulate CAD operation based on the same mathematical surface modelling (e.g. NURBS), and data I/O follows a universal data structure, e.g. IGES format. The two approaches offer advantage in different aspects, i.e. the former is more straightforward in implementation into design work, and the latter more independent from CAD system itself. The authors recently demonstrated both approaches, i.e. in Reference [1] and References [11, 12], respectively.

In the present study, the system 1 is used. Earlier version of the optimization module was demonstrated in Reference [1]. Optimization system parameters, e.g. number of design parameters, optimization algorithm, and definition of objective and constraint functions, can be specified in spreadsheet screen. See Reference [1] for more detailed features including



Figure 1. Basic elements of a CFD-based optimization environment.

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 2. Implementation of CAD-based hull-form modification into optimization environment. System 1—CAD direct control approach and System 2—CAD emulation approach, left and right, respectively.



Figure 3. Comparison of strategy in optimal search between SQP and GA approaches.

EXE/DLL file interfaces and the network-passage protocol. In the present study, new features based on MOGA algorithm for solution to multi-objective optimization problem are included. An overview of the present schemes is given in the following.

2.1. Optimization method

Two optimization algorithms are used in the present study, i.e. SQP [1-5] and MOGA [12, 26], whereas the former is mainly used to verify the results from the latter. Figure 3 illustrates differences in strategy between SQP and GA. SQP is able to efficiently search optimal if the initial point is correctly given. In contrast, GA is capable for global optimal search, and does not require evaluation of gradients. Another important feature of GA is that it can be extended to find Pareto-optimal solutions in multi-objective optimization. After the Pareto optimal is determined, a simple decision maker theory can be used to select final solution on the set. In the following, an overview of the present optimization scheme is given.

2.1.1. Single-objective optimizer—SQP approach. The present single-objective optimization method is based on SQP algorithm. A general expression of the single-objective optimization

Copyright © 2006 John Wiley & Sons, Ltd.

Int. J. Numer. Meth. Fluids 2006; 52:499-527

502

problem is defined as follows:

$$\operatorname{Min} |F(\boldsymbol{\beta}; Rn, Fn)|_{Rn, Fn} \tag{1}$$

s.t.
$$H_i(\beta) \ge 0$$
 $(i = 1, 2, ..., q)$ (2)

where $\beta = (\beta_1, \beta_2, ..., \beta_k)$ are design parameters, F is the objective function to be minimized, and H_i are inequality constraint functions. In SQP scheme, the objective and constraint functions are approximated in quadratic form such that

Min
$$[\nabla F(\boldsymbol{\beta}; Rn, Fn)^{\mathrm{T}} \mathbf{d} + \frac{1}{2} \mathbf{d}^{\mathrm{T}} B \mathbf{d}]_{Rn, Fn}$$
 (3)

s.t.
$$H_i(\boldsymbol{\beta}) + \nabla H_i(\boldsymbol{\beta})^{\mathrm{T}} \mathbf{d} = 0 \quad (i = 1, 2, ..., q)$$
 (4)

where $\mathbf{d} = (d_1, d_2, \dots, d_k)$ is the direction vector, and *B* is the approximate Hessian matrix of the Lagrangian. In each optimization cycle (*n*), optimum **d** is obtained so as to minimize *F*, and $\boldsymbol{\beta}$ is updated by $\boldsymbol{\beta}^{n+1} = \boldsymbol{\beta}^n + \mathbf{d}$. In the present study, the derivative terms in the above equations are evaluated by a second-order central finite difference scheme. Advantage of SQP over SLP (successive linear programming) was shown in the author's precursory work, and quite a few demonstrations with SQP were performed for optimization of tanker hull-form and naval surface combatant [1–5]. Reference [27] is recommended for more details of SQP.

2.1.2. Multi-objective optimizer—MOGA approach. Next, solution scheme for multi-objective optimization problem is described. A general expression of N-objective function optimization problem is defined as follows:

$$\operatorname{Min} \begin{cases} [F_1(\boldsymbol{\beta}; Rn, Fn)]_{Rn, Fn} \\ [F_2(\boldsymbol{\beta}; Rn, Fn)]_{Rn, Fn} \\ \vdots \\ [F_N(\boldsymbol{\beta}; Rn, Fn)]_{Rn, Fn} \end{cases}$$
(5)
s.t. $H_i(\boldsymbol{\beta}) \ge 0 \quad (i = 1, 2, ..., q)$ (6)

In the present study, the adopted scheme is an extended genetic algorithm (GA) [28-30] for multi-objective optimization problem, i.e. MOGA. The basic procedure follows that of GA: (i) generation of an initial population of individuals at random manner; (ii) decoding and evaluation of some predefined quality criterion, referred to as the fitness; (iii) selection of individuals based on a probability proportional to their relative fitness; (iv) crossover and mutation. The steps (ii)–(iv) are repeated until the generation achieves designated number.

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 4. Pareto ranking and sharing operations. Fitness is given based on Pareto ranking, and additive fitness is considered based on uniformity of individual distribution on Pareto-optimal set. These are used in MOGA.

The extension of GA for MOGA is straightforward. Main goal is to detect uniformly distributed globally Pareto-optimal front. Definition of globally Pareto-optimal set is as follows: the nondominated set of the entire feasible search space is the globally Pareto-optimal set [30]. In order to make the conditions of Pareto optimality mathematically rigorous, we state that a vector **x** is particularly less than **y**, symbolically $\mathbf{x} < P\mathbf{y}$ when the following condition holds: $(\mathbf{x} < P\mathbf{y}) \Leftrightarrow (\forall i)(x_i \leq y_i) \land (\exists i)(x_i < y_i)$. Under this circumstance, we say that point **x** dominates point **y**. If a point is not dominated by any other, we say that it is nondominated or noninferior. The basic definition is used to find noninferior points in MOGA in association with Pareto-ranking technique and sharing method in the present study. At each generation, higher fitness f_0 is given to individuals of higher Pareto ranking R_P , and at the same time, additive fitness f_s is given to individual with the best quality in one of objective functions (see Figure 4 for example of two-objective function case), i.e. $f_0 = 1/R_P + f_s$. The functional constraints are accounted for by using a penalty function approach, which artificially lowers the fitness if the constraints are violated and is expressed as

$$f = f_0 - r \left[\sum_{j=1}^{M} |h_j(x)| + \sum_{j=1}^{N} |\min\{0, -g_j(x)\}| \right]$$
(7)

At present, the GA and MOGA schemes can be used in both parallel and serial modes. The authors developed parallel-computing GA and MOGA by introducing message passing interface (MPI) protocol [31] (see Figure 5), and demonstrated single and multi-objective optimizations of naval surface combatant [11, 12]. It was shown that both GA and MOGA schemes yield satisfactory results. Introduction of parallel architecture effectively enhanced computational speed. Figure 6 shows results from the initial evaluation of the scheme, where sufficiently fast convergence towards analytically given Pareto-optimal front is indicated. In addition, the author and colleagues evaluated relative performance of the present MOGA scheme to others [26] by using more complex six algebraic test functions. Figure 7 shows some representative results from the work. The test functions T_1 (Equation (8), N = 2) and

Copyright © 2006 John Wiley & Sons, Ltd.

CFD-BASED MULTI-OBJECTIVE OPTIMIZATION METHOD



Figure 5. Serial and parallel architectures for GA optimizer. The conventional GA algorithm has been extended for high-performance optimization method by introducing parallel computing.



Figure 6. MOGA results. Determination of Pareto front and convergence of the individuals to the front for an algebraic test function. Sufficiently fast convergence is demonstrated for this test case.

 T_6 (Equation (9), N = 30) have a convex Pareto-optimal front, and a Pareto-optimal front consisting of several noncontiguous convex parts, respectively.

$$f_{1}(\bar{x}) = x_{1}$$

$$f_{2}(\bar{x}) = H(1 - \sqrt{f_{1}(\bar{x})/H})$$

$$f_{1}(\bar{x}) = x_{1}$$
(8)

$$f_2(\bar{x}) = H(1 - \sqrt{f_1(\bar{x})/H}) - f_1(\bar{x})\sin(10\pi f_1(\bar{x}))$$
(9)

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 7. Comparative results on the algebraic test functions. Test functions T_1 (N = 2) and T_6 (N = 30) have a convex Pareto front and a discrete Pareto front, respectively. Performance of the present MOGA approach is shown equivalent to that of MODPSO scheme.

where

$$H = 1 + \frac{9}{N-1} \sum_{i=2}^{N} x_i \tag{10}$$

Being focused on multi-objective problems with expensive objective functions, we decided to fix the maximum number of function evaluations to 100*N*. Bounds on the value of the design variables are also applied ($0 \le x_i \le 1$). As indicated in Figure 7, performance of the present MOGA is shown equivalent to that of one of the advanced deterministic optimization scheme, a multi-objective deterministic particle swarm optimization (MODPSO) scheme [26]. Reader may refer to Reference [30] for more various test problems.

2.2. Geometry modelling method

In ship design, a figure named *body plan* is used to show the shapes of sections determined by the intersection of the hull-form with planes perpendicular to the longitudinal axis. Figure 8 shows the body plan for ships used in the present study. As is the case for those ships, most ships are symmetrical about the centreplane, and the body plane shows only a half part of ship. In this paper, the centreplane intersection is referred to as the profile. Body plan stations are customarily numbered from stern to bow (particularly in European and Japanese shipyards) such that the fore and after maximum stations are S.10 (forward perpendicular-FP) and S.0 (after perpendicular—AP), respectively. Also, section at S.5 is called midship section. In the present study, the FP and AP, respectively, correspond to x=0 and 1 in computational coordinates. Another important hull-form characteristic in ship design is a prismatic curve (often referred to as C_p curve), which shows longitudinal distribution of sectional area of ship. Figure 9 shows an example of the curve. In many modern vessels, particularly cargo or tanker vessels, the form of cross-section below the design waterline extends without change for some distance in forebody an afterbody usually including midship location. Such vessels are said to have parallel middle body, which is also the case for the present ships discussed below. In the following, geometry modelling method used in the present study is described.

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 8. Comparison of body plan for SR221A and SR221B hulls. Left and right are SR221A and SR221B, respectively, while forebody is same between the two hulls. The blending parameters (T) are 0 and 1. Stern frame lines are more V and U types, respectively. Propulsive factors 1 + k and $1 - w_n$ are smaller and larger for A form.



Figure 9. Geometry of sectional area curve for ship. X = 0 and 1 correspond to FP (forward perpendicular, SS.10), and AP (after perpendicular, SS.0), respectively. X = 0.5 (SS.5) is called midship section. Sectional area is constant in parallel middle body.

2.2.1. CAD module (NAPA). The CAD module used in the present work is NAPA system [19]. As mentioned earlier, an important feature for the automatic optimization is the parametric expression and modification of the ship hull-form, which is apparently a key feature of NAPA system. Figure 10 shows an example for the parametric definition of SR221A tanker hull form, a basic hull-form used in the present study. The NAPA Macro is used to aid interfaces among CAD, optimizer, and CFD modules. In the NAPA Macro, the procedures of hull-form modification, geometry analysis, and CFD pre-processing are also described. Reference [1] includes a detailed example for the description. In combination with the CAD model, a practical hull-form modification method is proposed, i.e. that is based on one-parameter hull-form blending and two-parameter prismatic-curve control. These approaches are described in the following.

2.2.2. Hull-form blending (morphing) approach. New hull form is defined by blending of two (or more) basic hull-forms through blending parameter. This approach is also referred to as morphing. The basic hull-forms are defined by same numbers of control points. In the

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 10. Parametric expression of SR221A tanker hull form by NAPA system.



Figure 11. Parametric modification of prismatic curve (C_p curve).

present work, two-hull-form blending is considered by using SR221A and SR221B forms (shown in Figure 8). The operation is performed as follows:

$$\mathbf{P} = (1 - T)\mathbf{P}_{\mathrm{A}} + T\mathbf{P}_{\mathrm{B}} \tag{11}$$

where \mathbf{P} , \mathbf{P}_{A} , and \mathbf{P}_{B} are control points for new hull form, hull A, and hull B, respectively; and T is blending parameter. This approach can directly be implemented into ongoing design work. As is demonstrated later, of practical interest is to find the best compromise between the two hulls with opposite hydrodynamic features in design tradeoff.

2.2.3. Prismatic curve parametric modification. New hull-form is directly correlated with parametric modification of prismatic curve (C_p curve). This idea is an extension of an original scheme used in many shipyards [20]. Figure 11 shows definition of the control parameters, where the area distribution is normalized by that of midship section (x = 0.5) for convenience. In the figure, new C_p curve is parametrically given by ΔC_p and ΔL_p , through definition of longitudinal movement of sections, i.e. Δx , which is given by

$$\Delta x = (1-x) \left\{ \frac{\Delta L_p}{1-L_p} + \frac{x-L_p}{A} \left[\Delta C_p - \Delta L_p \frac{1-C_p}{1-L_p} \right] \right\}$$
(12)

Copyright © 2006 John Wiley & Sons, Ltd.

CFD-BASED MULTI-OBJECTIVE OPTIMIZATION METHOD



Figure 12. Overview of computational grid. Both port and starboard sides are included in order to account for asymmetric flow due to propeller action.

in association with the following definition of variables:

$$A \equiv C_p (1 - 2\bar{x}) - L_p (1 - C_p) \tag{13}$$

and

$$C_p \cdot \bar{x} \equiv \frac{1}{2} \int_0^1 x^2 \,\mathrm{d}y, \quad C_p \equiv \int_0^1 x \,\mathrm{d}y \quad \text{and} \quad \Delta C_p = \int_0^1 \Delta x \,\mathrm{d}y$$
 (14)

In the present study, this approach is used in combination with the above-mentioned hull-form blending approach.

2.3. CAD-interfaced automatic grid generator

In the present work, a recently developed CAD-interfaced automatic grid generator is applied. Surface as well as volume grids are automatically generated based on prescribed set-up parameters (which are basically same in series case studies if grid topology is fixed). The volume grid is generated by an elliptic-algebraic method using an exponential scheme, and method of lines. A concern for automatic gridding will be robustness to practical complexities of hull surface. Through preliminarily exercises, the present scheme was shown capable for application to tanker hull-forms, surface combatants, and container ships [32]. In all cases, the grid orthogonality especially near the hull surface is sufficiently maintained. Figure 12 shows several views of the present computational grid, which is categorized as O–O-type topology. The grid includes both port and starboard sides for self-propulsion condition in order to simulate asymmetric flows due to influences of propeller action.

2.4. CFD module (RaNS equation solver—FLOWPACK Version 2004d)

The RaNS code is FLOWPACK version 2004d, which has been developed by the authors for CFD education and research, and design applications for ship hydrodynamics, aerodynamic and fluid engineering [21–23]. In the transition for design applications, multi-block domain decomposition capability is included. At present, FLOWPACK has tight interface with both commercial and author's in-house grid generators. Summary of applications is available in

Copyright © 2006 John Wiley & Sons, Ltd.

Int. J. Numer. Meth. Fluids 2006; 52:499-527

509

Reference [22] for full-scale simulation, hull-form optimization, America's Cup down-wind sail system, fully appended sailing boat, parachute, automobiles and others.

The numerical method of FLOWPACK solves the unsteady RaNS and continuity equations with zero or two-equation turbulence model for mean-velocity, pressure and eddy viscosity or turbulence parameters by using a body/free-surface conforming grid. The equations are transformed from Cartesian coordinates in the physical domain to numerically generated, boundary fitted, nonorthogonal, curvilinear coordinates in the computational domain. A partial transformation is used, i.e. coordinates but not velocity components. The equations are solved using a regular grid, finite-analytic spatial and first-order backward difference temporal discretization, PISO-type pressure algorithm, and the method of lines. FLOWPACK is able to consider wavemaking effects by using free-surface tracking approach; however in the present work, the feature is not used.

2.5. Self-propulsion simulator

In the present study, we consider a ship propelled by a single propeller located near the stern. It is generally observed that the propeller when developing thrust accelerates the water ahead of it, and this has effect of lowering the pressure around the stern and also increasing the velocity there, both of which effects augment the resistance of the ship. This is called thrust deduction, which will be quantitatively defined below and in nomenclature. In the present study, the propulsive performance is represented by the DHP, which is the power actually used for rotating propeller and one of the most important design parameters in ship design. Another important propulsive factor is the effective wake, which is defined by subtracting propeller-induced velocity from total velocity at the propeller disk. This directly correlates with propeller performance, and is obviously different with the ship wake without propeller-induced effects. The latter is often called the nominal wake in contrast to the effective wake. The wake parameters will be used in the subsequent sections where the results are discussed.

The ship propulsive performance is evaluated by the present self-propulsion scheme [23]. The method consists of three parts, i.e. a RaNS solver, a propeller performance program, and a root finder module to determine propeller rotational speed n_p so that the hull resistance R balances with the propeller thrust T, i.e.

$$T = R \tag{15}$$

In the model scale propulsion test, which is usually carried out in the towing tank during a ship design, difference in skin friction between the model and full-scale ships must be considered as a skin-friction correction, i.e. SFC. This eventually reduces the thrust required to balance with the hull resistance, i.e.

$$R = R_{\rm T(SP)} - \rm SFC \tag{16}$$

where $R_{T(SP)}$ is the hull resistance for self-propulsion state. In the present study, the SFC is given by using C_{F0M} , C_{F0S} , and ΔC_F , all of which are values for frictional resistances for model and full-scale and for consideration of scale effects and surface roughness, i.e.

$$SFC = \frac{\rho U^2 S_0}{2} \{ (1+k)(C_{F0M} - C_{F0S}) - \Delta C_F \}$$
(17)

Copyright © 2006 John Wiley & Sons, Ltd.

where subscripts M and S correspond to model and full-scales, respectively. In the present work, these follow values used in SR229 research project described later. Propulsive factors are evaluated when the self-propulsion state is achieved. For example, the thrust deduction factor 1 - t and DHP are given by

$$1 - t = \frac{R_{\mathrm{T(Tow)}} - \mathrm{SFC}}{T} = \frac{R_{\mathrm{T(Tow)}} - \mathrm{SFC}}{R_{\mathrm{T(SP)}} - \mathrm{SFC}}$$
(18)

and

$$DHP = \frac{2\pi n_p Q}{75}$$
(19)

where $R_{T(Tow)}$ is the hull resistance for towing state, and Q is propeller torque. Propeller action effects are included in RaNS equations by the body force approach. The body force distribution is interactively and iteratively determined by propeller performance calculation based on infinitely bladed propeller theory [24], by using free vortex distribution in wake and bound vortex distribution on the propeller disk. The disk is divided into $36 \times 5 = 180$ panels in the present work. Figure 13 shows the overall computational procedure of the present



Figure 13. The present self-propulsion scheme. Propeller performance calculation is based on infinitely bladed propeller theory. Root finder determines propeller rotational speed n_p so that propeller thrust balances with hull resistance. Thrust deduction and SFC are included for ship-point test.

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 14. Definition of coordinate system for the present maneuvering simulator.

self-propulsion scheme. Note that flows for self-propulsion condition are computed by using those for towing condition as initial guess. In the present applications, numbers of global sweep iteration in RaNS solver are normally 3000 and 2000 for towing and self-propulsion conditions, respectively. Finally, the DHP directly obtained from the present self-propulsion scheme is used as one of multi-objective functions to be minimized.

2.6. Ship maneuvering simulator

The ship maneuverability performance, which is another multi-objective function, is evaluated by using the present ship maneuvering simulator. The scheme was recently developed by the present authors. The method solves the following equation of ship motions for surge, sway, and yaw motions in time-marching manner (see Figure 14 for definition of coordinate system):

$$F_X = (m + m_x)(\dot{U}\cos\beta - U\dot{\beta}\sin\beta) + (m + m_y)U\omega\sin\beta$$
$$F_Y = -(m + m_x)(\dot{U}\sin\beta + U\dot{\beta}\cos\beta) + (m + m_y)U\omega\cos\beta$$
$$N = (I_{ZZ} + i_{ZZ})\dot{\omega}$$
(20)

where m, m_x, m_y , are mass and added masses; I_{ZZ} and i_{zz} are vertical-axial moment and added moment; F_X and F_Y are axial hydrodynamic forces; β and ω are yaw angle and verticalaxial angular velocity; U is ship speed; and dot indicates time derivative. The forces are defined in ship-fixed coordinates X-Y, which are independent from ground-fixed coordinate X_g-Y_g . The hydrodynamic forces are given by the method of Kijima and Nakiri [25] including extensions for tanker hull forms. In the present study, the 10/10 degree-Zigzag test is simulated and the first OSA is used as one of multi-objective functions to be minimized. As shown in Figure 15, the present scheme successfully predicts the trends shown in the measurements for basic hull forms SR221A, SR221B, and SR221C [25]. In the demonstration of optimization, computational values are corrected through the correlation curve shown in the figure, which enables to provide quantitatively more accurate predictions.

Copyright © 2006 John Wiley & Sons, Ltd.

CFD-BASED MULTI-OBJECTIVE OPTIMIZATION METHOD



Figure 15. Comparison of the first OSA in 10/10-degree-Zigzag test. Computational results are obtained by using the present ship maneuvering simulator. Trends shown in experiments are correctly reproduced by the present predictions.

3. UNCERTAINTY ASSESSMENT AND ACCURACY IN PROPULSIVE FACTORS AND FLOWS

3.1. Uncertainty assessment

Using the CFD methods, uncertainty assessment must be provided for the solutions and computational grid. CFD uncertainty assessment consists of verification, validation, and documentation. Simulation uncertainty $U_{\rm S}$ is divided into two components, one from numerics $U_{\rm SN}$ and the other from modelling $U_{\rm SM}$. The $U_{\rm SN}$ is estimated for both point and integral quantities and is based upon grid and iteration studies which determine grid U_{SG} and iterative $U_{\rm SI}$ uncertainties. A root sum square (RSS) approach is used to combine the components and to calculate $U_{\rm SN}$, i.e. $U_{\rm SN}^2 = U_{\rm SG}^2 + U_{\rm SI}^2$. CFD validation follows the method of Stern et al. [33] and Wilson et al. [34], in which a new approach is developed where uncertainties from both the simulation $(U_{\rm S})$ and EFD benchmark data $(U_{\rm D})$ are considered. The first step is to calculate the comparison error E which is defined as the difference between the data D(benchmark) and the simulation prediction S, i.e. E = D - S. The validation uncertainty U_V is defined as the combination of $U_{\rm D}$ and the portion of the uncertainties in the CFD simulation that are due to numerics $U_{\rm SN}$ and which can be estimated through verification analysis, i.e. $U_{\rm V}^2 = U_{\rm D}^2 + U_{\rm SN}^2$. $U_{\rm V}$ sets the level at which the validation can be achieved. The criterion for validation is that |E| must be less than U_V . Note that for an analytical benchmark, U_D is zero and $U_{\rm V}$ is equal to $U_{\rm SN}$. Validation is critical for making improvements and/or comparisons of different models since $U_{\rm SN}$ is buried in $U_{\rm V}$.

The above mentioned were applied to evaluate the present CFD method. Table I shows uncertainties and errors for total resistance for KRISO Container Ship (KCS) towing condition test case (Fn = 0.26 and $Rn = 1.4 \times 10^7$) [23]. The size of computational grids is about 250 000 and smaller grid is prepared by using refinement ratio $r = \sqrt{2}$, i.e. around 900 000. Order of

Copyright © 2006 John Wiley & Sons, Ltd.
Uncertainties U _D (%D)	<i>U</i> _G (%D)	<i>U</i> _I (%D)	U _V (%D)
1.0	2.0	0.2	2.2
Errors CFD (S) 0.00355	EFD (D) 0.00356	<i>E</i> (%D) 0.42	

Table I. Uncertainties and errors for total resistance.

KCS test case for towing condition. Fn = 0.26 and $Rn = 1.4 \times 10^7$.

	$C_{\mathrm{T(Tow)}}$	$1 - w_n$	$C_{\mathrm{T(SP)}}$	K_T	KQ
CFD	0.00355	0.634	0.00393	0.1670	0.0282
EFD	0.00356	0.686		0.1700	0.0288
	1-t	$1 - w_{\mathrm{T}}$	η_0	$\eta_{ m R}$	η
CFD	0.8515	0.789	0.631	1.074	0.732
EFD	0.8530	0.792	0.682	1.011	0.740
	J	$n_{\rm P}~({\rm rps})$			
CFD	0.718	9.528			
EFD	0.728	9.500			

Table II. Comparison of propulsive factors.

KCS test case for self-propulsion condition. Fn = 0.26 and $Rn = 1.4 \times 10^7$.

accuracy P_G is 1.7, which is given by the previous experience, and the correction factor is given as $C_G = 0.8$. For $C_G = 0.8$ considered as sufficiently less than 1 and lacking confidence, $U_G = 2.0\%$ D is estimated. The variation in the total resistance is 0.2%D over the last period of oscillation, i.e. $U_I = 0.2\%$ D. Finally, $U_D = 1\%$ D and $U_{SN} = 2\%$ D yield $U_V = 2.2\%$ D. It is shown that the CFD result is validated for the indicated U_V level. In authors' judgment, the agreement between CFD and EFD (experimental fluid dynamics) results is satisfactory for this level of grid size, which is due to a fact that other CFD results recently presented for this test case are generally overestimated and the variation of data was about 5% [32, 35]. Further discussion on propulsive factors will be made in the following section.

3.2. Accuracy in propulsive factors and flows

Table II shows comparison of propulsive factors between CFD (i.e. the present self-propulsion simulator) and EFD results for KCS self-propulsion condition test case (Fn = 0.26 and $Rn = 1.4 \times 10^7$) [23]. It is shown that generally good agreement is demonstrated. Due to action of propeller, the hull resistance increases about 10%. The increase in hull resistance is mainly due to increase of pressure resistance. Frictional resistance also increases, but the magnitude is insignificant. Influences of propeller action on flows are also correctly reproduced by the present numerical scheme. These are drastic especially in the region right after the propeller. The flow exhibits characteristics of an asymmetric swirling jet and accelerated velocity

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 16. Effective wake, propeller-induced axial velocity, and axial body force contours on propeller disk for with-propeller condition $(Rn = 1.4 \times 10^7, Fn = 0.26)$: (a) effective wake; (b) propeller induced axial velocity; and (c) axial body force. KCS test case for self-propulsion condition.



Figure 17. Comparison of surface pressure (C_p) contours between without/with-propeller conditions $(R_n = 1.4 \times 10^7, F_n = 0.26)$. KCS test cases. Notations 'without/with-propeller conditions' correspond to towing and self-propulsion conditions, respectively.

fields, and importantly, larger acceleration of flow in starboard side than the other side for the given propeller rotational direction (clockwise direction). The present CFD successfully captures such general features of flow [23]. Figure 16 shows the effective wake, propeller-induced axial velocity, and axial body force contours on the propeller disk. Note that effective wake u_e is defined as $u_e = u_t - u_p$, where u_t and u_p are the total- and propeller-induced axial velocity, respectively. Due to the propeller–hull interaction, asymmetric distribution of those values is evident. The axial body force directly correlates with u_p and flows just downstream of the propeller, where the flow is dominated by the propeller-induced effects.

Figure 17 shows comparison of surface pressure contours between towing and selfpropulsion cases (without prop. and with prop. in the figure, respectively). The latter case

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 18. Comparison of 1 + k and $1 - w_n$ between computation and experimental data. SR221B tanker hull form for towing condition.

clearly exhibits expected decrease due to the action of the propeller, which is more evident in the region the boundary layer is thick, i.e. the region near the stern bulb and near the propeller. It is also shown that the influence of the propeller is restricted to the stern region, as is commonly assumed in hull-form design. The trends are also shown in EFD data as backward shift of pressure contours, which agrees well with that predicted by CFD [23].

Accuracy in prediction of resistance and propulsive factors is also evaluated for tanker hull form. The wavemaking effects are not considered. In general, wavemaking effects are neglected in simulation of flow for this type of ship hull, since Froude number of the design speed is generally low, e.g. Fn = 0.15. This leads to a fact that smaller number of computational grids than the previous case can be used if the main interest is placed on only boundary-layer and wake flows and hydrodynamic forces. Figure 18 shows comparison of form factor (1+k) and nominal wake coefficient $(1 - w_n)$ between CFD and EFD results. Although relatively small size grid was used, i.e. 500 000, accuracy in the values is nearly that of experimental data.

3.3. Introduction of CFD-EFD correlation curve for optimization

The above-discussed supports the validity of the present numerical scheme for not only qualitative but also quantitative accuracy if appropriate size of computational grid is used. In reality, CFD-based optimization is time-consuming and size of computational grid is often limited to those which can complete the task within allowable hours. Since a decrease in grid size may yield an increase in simulation uncertainty, it will be difficult to produce results with meaningful improvements if CFD predictions are directly applied.

However, if an optimization aims to search for the best design trade-off and the variation of feasible design modification is relatively limited, the indicated problem can be solved by an alternative approach. Here, we introduced CFD–EFD correlation curves where the correlations between the present CFD and EFD data are examined for basic hull forms SR221A, SR221B and SR221C using a relatively small grid around 200 000. As demonstrated later, the optimal solution is obtained using those basic hull forms. In addition, in order to further reduce computational loads, Baldwin–Lomax model [36] was used in the current study as the algebraic turbulence model. As shown in Figure 19, Results indicate that our computations correctly reproduced the trends in measurements. The same grid size and turbulence model are used through the optimization, where CFD predictions are always corrected using the CFD–EFD correlation curves shown in the figure. The values are considered to be the equivalent of EFD predictions and are therefore used for the evaluation of optimal results.

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 19. Comparison of propulsive factors between computations and experiments. Trends shown in experiments are correctly reproduced by the present predictions.

4. TANKER STERN FORM OPTIMIZATION PROBLEM

4.1. Hydrodynamic aspects in design optimization

In the determination of tanker stern form, hull-form designers must find the best compromise between the so-called V- and U-shaped stern cross-sections. V-shaped sterns induce weaker stern bilge vortices and lower viscous resistance. In contrast, U-shaped sterns result in generation of stronger vortices and higher viscous resistance; however, the propeller inflow is more uniform with better cavitation characteristics and lower noise. If DHP minimization is performed, flow features to be considered are more comprehensive, i.e. lower hull resistance, better propeller inflow and thrust deduction, all of which are combined effects of propeller– hull interactions. The design tradeoff also occurs when the ship maneuverability is accounted

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 20. Comparison of geometry between the original and optimized tanker hull forms. Optimized hulls in (a) and (b) are VR-Min. and DHP-Min., respectively.

for, i.e. U-shaped sterns result in better maneuverability performance, e.g. the smaller OSAs in zigzag motions.

In authors' precursory work for tanker stern optimization [4], it was shown that minimum DHP hull form does not coincide with minimum viscous resistance hull form (those are referred to as DHP-Min. and VR-Min., respectively). Earlier version single-objective optimization scheme, CFD and propeller model, and geometry modelling method were used for SR221B stern optimization. Figure 20 shows comparison of geometry between the original and optimal hull forms. Differences in stern frame lines are obvious between the original and optimized hull forms, in which clear differences between DHP-Min. and VR-Min. are also indicated, i.e. frame line modification from U to V types is significant for the latter, in contrast, the trend is somewhat reduced for the former. From U to V frame line modification of VR-Min. results in lower viscous resistance; however, also larger effective wake $(1 - w_i)$ which causes lower propeller efficiency and larger DHP. This implies an important fact that hull-form optimization for minimum viscous resistance does not yield minimum

Copyright © 2006 John Wiley & Sons, Ltd.

delivered-horsepower hull form. All of the above-mentioned modification trends agree well with those commonly in use in traditional tanker hull-form design.

4.2. Definition of the present optimization problem

For the complete definition of the design problem to be solved, the following fundamental items must be precisely addressed: (1) selection of an initial design to be optimized and of the extension of the modifiable region; (2) choice of the objective function to be minimized plus number and position of the design variables; and (3) type and quantity of the constraints of the problem. All these items are described in the following.

4.2.1. Initial design. The initial design hull forms are SR221A and SR221B tankers, which were conceived as preliminary designs for VLCC tanker hull forms for extreme V- and U-type sterns, respectively, and selected in Domestic Japan Research Projects ca. 1990. Additionally, SR221C form is examined in the projects as an in-between the two hull forms. All hull forms have the same forepart. All include bulbous bow and stern bulb and are apparently more modern designs than earlier test models SR196 or HSVA tankers. Propulsion is provided through single open-water propeller. There is a large EFD database for SR221 series tankers due to the collaborative study on EFD/CFD and hull-form optimization among universities and shipbuilding industries (SR229, April 1996–March 1999) [37]. The EFD data include measurements on stern flows, propulsive factors, and maneuverability performance. In the present study, bare-hull stern optimization (after midship) is considered including propeller influences, in which the propeller model and the POT characteristics follow those studied in SR229.

4.2.2. Objective function, and functional and geometrical constraints. Complete definition of the problem, objective function and constraints, is given in Table III. For the present demonstration, free-surface effects are not considered. Two objective functions are considered, i.e. (i) the DHP at a speed of $Rn = 3.11 \times 10^6$ at model scale, and (ii) first OSA for 10/10degree-Zigzag test, all of which are simultaneously minimized. Design *Fn* is set to 0, since for this hull-form wavemaking effects are negligibly small. Geometrical constraints are imposed on the profile, the design variables, and the displacement and principal dimensions of the ship. In addition, static sinkage and trim conditions are used, which are apparently simplification of the practical optimization problem but justified due to main objective of the present work, i.e. system development and demonstration. More details are given in Table III.

4.2.3. Design variables definition. The design variables are used to explore the design space, and changes in their values correspond to different ship design. As a consequence, those are closely connected with the specific technique adopted to modify the geometry of the ship and the computational mesh, i.e. the geometry manipulation method which is implemented in the overall optimization scheme. Hence, the design variables in the present work are parameters in hull-form blending and C_p curve control, i.e. T in Equation (11), and ΔC_p and ΔL_p in Equation (12). Table III shows constraints for the variables.

Copyright © 2006 John Wiley & Sons, Ltd.

Y. TAHARA, S. TOHYAMA AND T. KATSUI

Туре	Definition	Note
<i>Objective functions</i> 2 objective functions: propulsive and maneuvering performances	$F_1 = \text{DHP} (\boldsymbol{\beta}; Rn, Fn)$	Delivered horsepower at ship-point self-propulsion condition (DHP) and first overshoot angle at 10/10 Z-test (OSA) for $Rn = 3.11 \times 10^6$
	$F_2 = \text{FOA} \ (\boldsymbol{\beta}; Rn, Fn)$ $\boldsymbol{\beta} \in R^{N_{dv}}$	No free-surface effects, i.e. $Fn = 0$
Geometrical constraints		
Profile	Profile is fixed	
Variation of	$0 \leqslant T \leqslant 1$	T: Blending parameter
design variables	$-1 \leqslant dCp \leqslant 1$ $-1 \leqslant dLp \leqslant 1$	dC_p and dL_p : C_p curve modification parameters Bare hull, fixed model. Afterbody (after midship) is optimized
Main dimensions	$L_{\rm pp}$, <i>D</i> , and <i>B</i> are fixed Forebody is fixed	
Displacement	$\frac{\Delta}{\Delta_{\text{Original}(221B)}} \ge 0.998$	Minimum displacement is 99.8% Δ of SR221B

Table III. Definition of objective functions and constraints.

5. RESULTS AND DISCUSSION

5.1. Multi-objective optimization environment

The present multi-objective optimization for minimum DHP–OSA was performed on a single CPU Personal Computer (Pentium 4, 2.6 GHz). The system parameters of MOGA are as follows: crossover rate = 1.0, mutation rate = 0.02, population size = 10, and number of maximum generation = 10. For the present multi-objective optimization, Pareto-optimal front obtained in 10 generations is represented as a final solution. Wall-clock time to proceed 10 generations was about 100 h, i.e. about 4 days, which is considered to be within practical turnaround. The time will be considerably shortened by using parallel computing environment [31], but that was not considered in the present work. As already noted, SQP is used for single-objective optimization for minimum DHP, where in this case the convergence criterion was satisfied at eight global optimization cycles. The results are used for verification of multi-objective optimization as described below.

5.2. Optimized hulls

Figure 21 shows Pareto front obtained in 10 generations for multi-objective optimization. In the figure, the individuals which violate design constraints are excluded. Figures 22 and 23 show comparison of body plan, surface pressure and streamlines for the present optimal hull forms. In addition, Tables IV and V show summary of properties and propulsive factors. In the figures and tables, results for original (or basic) hull-forms, i.e. SR221A, SR221B, and SR221C, are also included for comparison. In the tables, all values are expressed in relative% differences to values for SR221A (hereafter, referred to as S_A). In the demonstration of optimization and discussions to follow, computational values are corrected through the correlation curves shown in Figures 15 and 19, which enables to provide quantitatively more accurate

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 21. Pareto front obtained in 10 generations for multi-objective optimization.



Figure 22. Comparison of bodyplan, surface pressure, and streamlines between the original and single-objective (DHP) minimal hull forms. Shaded area in pressure contours indicates pressure pocket region.

predictions. Notations S-Opt and M-Opt correspond to results for single- and multi-objective optimizations, respectively. It is important to note that all individuals on the Pareto-optimal front in Figure 21 will be candidates for designer's choice. Next task of designers is selection of final candidates, that may be based on designers' own multi-criteria for hydrodynamic performances possibly including other factors in ship-building process. The designer could use a decision-making technique to pick up one final solution among a set of Pareto-optimal

Copyright © 2006 John Wiley & Sons, Ltd.



Figure 23. Comparison of bodyplan, surface pressure, and streamlines among optimal hull forms. Shaded area in pressure contours indicates pressure pocket region.

Table	IV.	Properties	and	propulsive	factors of	of	original	and	single-ol	ojective	optimal	hulls.
							0		0	3	1	

	Wet. surface (%)	Displacement (%)	$R_{T(Tow)}$ (%)	$1 - w_t$ (%)	DHP (%)
A	0	0	0	0	0
В	-1.21	-0.33	+11.24	-39.52	+5.25
С	-0.86	-0.15	+3.17	-21.29	+2.50
S-Opt.	-0.17	+0.55	-4.50	+7.58	-3.13

 * %S_A - 100.

Table V. Properties and propulsive factors of original and multi-objective optimal hulls.*

	Wet. surface (%)	Displacement (%)	$R_{T(Tow)}$ (%)	$1 - w_t$ (%)	1 - t (%)	DHP (%)	OSA (%)
A	0	0	0	0	0	0	0
S-Opt.	-0.17	+0.55	-4.50	+7.58	+1.00	-3.13	+40
M-Opt.01	-0.35	+0.59	+3.25	-14.49	+0.21	-2.25	-17
M-Opt.02	-0.78	-0.14	+4.22	-15.96	+0.86	+1.65	-56
M-Opt.03	-0.94	-0.51	+9.15	-31.77	-0.95	+7.10	-89

 * %S_A - 100.

designs. Some examples of the decision-making theory are described in Reference [30]. In the present study, three representative individuals on the Pareto front are selected, namely M-Opt.01–03, and used to discuss the trends in hull forms and flows.

5.3. Numerical results and verification

First, values in Table IV, i.e. those for basic hull-forms, are discussed. Wetted surface areas of basic hull forms SR221B and SR221C are smaller than that of SR221A, i.e. $-1.21\%S_A$ and $-0.86\%S_A$, respectively, and so are displacements, i.e. $-0.33\%S_A$ and $-0.15\%S_A$, respectively. Total resistances for towing condition $R_{T(Tow)}$ are larger, i.e. $+11.24\%S_A$ and $+3.17\%S_A$

Copyright © 2006 John Wiley & Sons, Ltd.

for SR221B and SR221C, respectively; but effective wakes $(1 - w_T)$ indicate reverse trends, i.e. $-39.52\%S_A$ and $-21.29\%S_A$, respectively. Resultant DHPs are larger, i.e. $+5.25\%S_A$ and $+2.50\%S_A$, respectively. As mentioned earlier, all of the predicted trends agree well with the measurements.

Next, the above discussions are continued for the optimal hulls (see Table V). Wetted surface areas of S-Opt, M-Opt.01, M-Opt.02, and M-Opt.03 are all smaller than that of SR221A, i.e. $-0.17\%S_A$, $-0.35\%S_A$, $-0.78\%S_A$, and $-0.94\%S_A$, respectively; and displacements are larger for S-Opt and M-Opt.01, i.e. +0.55%S_A and +0.59%S_A, respectively, and smaller for M-Opt.02, and M-Opt.03, i.e. $-0.14\%S_A$ and $-0.51\%S_A$, respectively. Thrust deductions (1 - t) are larger for S-Opt, M-Opt.01, and M-Opt.02, i.e. $+1.00\%S_A$, $+0.21\%S_A$, and +0.86%S_A, respectively; and smaller for M-Opt.03, i.e. -0.95%S_A. $R_{T(Tow)}$ and $(1 - w_T)$ indicate reverse trends, i.e. $-4.50\%S_A$, $+3.25\%S_A$, $+4.22\%S_A$, and $+9.15\%S_A$ for $R_{T(Tow)}$ of S-Opt, M-Opt.01, M-Opt.02, and M-Opt.03, respectively; and +7.58%S_A, -14.49%S_A, -15.96%S_A, and -31.77%S_A for $(1 - w_T)$. DHP and OSA also show reverse trends, i.e. -3.13%S_A, -2.25%S_A, +1.65%S_A, and +7.10%S_A for DHP of S-Opt, M-Opt.01, M-Opt.02, and M-Opt.03, respectively, and $+40\% S_A$, $-17\% S_A$, $-56\% S_A$, and $-89\% S_A$ for OSA. An important fact shown in the present results again supports a conclusion of the author's precursory work [4], i.e. the minimum $R_{T(Tow)}$ hull form does not coincide with minimum DHP hull form, and inclusion of propeller-hull interaction is necessary for meaningful DHP minimization.

It is an expected trend that S-Opt indicates the lowest DHP but the largest OSA. S-Opt has more enhanced V-type stern, and backward movement of volume distribution is seen in. The modification trends yield worsening of effective wake $(1-w_t)$ and thrust deduction (1-t), but lowering of the hull resistance which compensates for the worsening of effective wake and thrust deduction; and finally the combined effects of those result in the lowest DHP among all optimal hulls. The present MOGA finds relatively close neighbours of S-Opt. As the maximum number of generation increases, the closer point to S-Opt will be detected, which is due to the probabilistic nature of the MOGA theory. On the other hand, the trends shown in the sequence of S-Opt, M-Opt.01, M-Opt.02, and M-Opt.03 are found to change in sterns from V to U type. As DHP increases, OSA decreases. More details on hydrodynamic characteristics of those optimal hulls are discussed below.

For all hulls shown in the figures, flows near the stern are closely correlated with surface pressure distributions. The differences are clear for surface pressure distributions and limiting streamlines near the stern. Surface pressure distributions for all hulls indicate pressure pockets near the stern bilge, and those of the U-type stern tend to indicate larger low-pressure region and lower values near the centre of the region. The pressure pockets attract the flows from the side and bottom of the hull, that leads the merging of the flows. The deeper pressure pocket has stronger influences on the flows approaching towards stern, and leads three-dimensional flow separation and generation of stern bilge vortices. As the stern form changes from U to V types, surface pressure indicates smaller region of the three-dimensional flow separation, therefore produces weaker stern bilge vortices and lower viscous resistance. On the other hand, stronger stern bilge vortices yield lower effective wake, which is an advantage regarding propeller performance. Besides, extreme V-type sterns tend to results in worsening of thrust deduction and maneuverability performance. All of the above-mentioned trends in flow features are consistent with trends shown in the present optimal hulls.

Copyright © 2006 John Wiley & Sons, Ltd.

Y. TAHARA, S. TOHYAMA AND T. KATSUI

As shown in the results, conditions of Pareto optimality are satisfied, so that noninferior points are successfully found by excluding the points \mathbf{x} dominate point \mathbf{y} . Besides, optimal hulls on Pareto front are found superior to all initial hull forms. Moreover, the convex Pareto front includes reasonable number of final selections in spite of relatively small population size. These support validity of the present MOGA scheme in association with validity of the present problem definition, where opposite hydrodynamic characteristics in design tradeoff are minimized as multi-objective functions, i.e. DHP and OSA. In summary, results obtained from the present multi-objective optimization appear to be meaningful and very promising, which leads to a conclusion that approach developed and demonstrated in the present work warrants further investigation and extension for more capable CFD-based multi-objective optimization method in practical and productive hull-form designs.

6. CONCLUSIONS

This paper concerns development and demonstration of a CFD-based multi-objective optimization method for ship design. Three main components of the method, i.e. CAD, CFD, and optimizer modules are functionally independent and replaceable. The CAD used in the present study is NAPA system. The CFD method, FLOWPACK version 2004d, is implemented into a self-propulsion simulator, where the RaNS solver is coupled with a propeller-performance program based on infinitely bladed propeller theory in an interactive and iterative manner. In addition, a maneuvering simulation model is developed and applied to predict ship maneuverability performance. The system demonstration is carried out to simultaneously minimize delivered horsepower and the first overshoot angle obtained from the self-propulsion simulator and maneuvering simulation method, respectively. Two nonlinear optimization algorithms are used in the present study, i.e. the successive quadratic programming and the MOGA, while the former is mainly used to verify the results from the latter. In combination with CAD model, a practical hull-form modification method is proposed, i.e. that is based on one-parameter hull-form blending and two-parameter C_p -curve control.

As mentioned earlier, the present work was motivated to overcome limitations appear in most of recent studies related to CFD-based hull-form optimization. The shortcomings are attributed to limitations of a simple geometry modelling and capability of optimization scheme which basically follows single-objective optimization theory. For development of more advanced CFD-based optimization method for practical hull-form design, there were inevitable challenges for introduction of CAD-based geometry modelling scheme and optimization theory for multi-objective optimization problem, in which improvement of a specific aspect of the global design usually causes the worsening for some others. As shown in the present results, the present CAD-based geometry modelling successfully avoids unrealistic hull-form modification and the method can directly be implemented into ongoing design process. Moreover, results obtained from the present multi-objective optimization appear to be meaningful and very promising. All lead to a conclusion that approach developed and demonstrated in the present work is very promising and worthy to further investigate and extend for more capable CFD-based multi-objective optimization method in practical and productive hull-form designs. Extension of the problem and future research direction will involve application to high-speed ship in association with development and adoption of more advanced-level global optimization (GO) algorithms and CFD methods.

Copyright © 2006 John Wiley & Sons, Ltd.

Int. J. Numer. Meth. Fluids 2006; 52:499-527

524

CFD-BASED MULTI-OBJECTIVE OPTIMIZATION METHOD

NOMENCLATURE

<i>x</i> , <i>y</i> , <i>z</i>	non-dimensional Cartesian coordinates, normalized by ship length
	L _{pp}
u, v, w	velocity components, normalized by ship speed U_0
U_0	ship speed
ho	density of water
$Fn = U_0 / \sqrt{g \mathrm{L}_{\mathrm{pp}}}$	Froude number
$Rn = U_0 L_{pp} / v$	Reynolds number
g	gravitational acceleration
ν	kinematic viscosity
S_0	wetted surface area at rest
$C_{\rm T} = R_{\rm T} / \frac{1}{2} \rho U_0^2 S_0$	total resistance coefficient, where $R_{\rm T}$ is total resistance
$C_{\rm F} = R_{\rm F} / rac{1}{2} ho U_0^2 S_0$	frictional resistance coefficient, where $R_{\rm F}$ is frictional resistance
$C_{\rm P} = R_{\rm P} / \frac{1}{2} \rho U_0^2 S_0$	pressure resistance coefficient, where R_P is pressure resistance
$I + k = \tilde{C}_{\mathrm{T}}/C_{\mathrm{F0}}$	form factor
$C_{ m F0}$	ITTC 1957 frictional coefficient line, $C_{\rm F0}=0.75/(\log_{10} Re-2)^2$
$\Delta C_{ m F}$	roughness allowance
D	diameter of propeller
$d_{\rm h}$	diameter of a propeller hub, $d_h = D \times (hub ratio)$
$J=v_{\mathrm{a}}/n_{\mathrm{P}}D$	advance ratio
$K_{\rm T} = T/\rho n_{\rm P}^2 D^4$	thrust coefficient
$K_Q = Q/\rho n_{ m P}^2 D^5$	torque coefficient
$K_{Q(O)}$	torque coefficient in open water (uniform flow)
n _P	propeller rate of revolution (rps)
\mathcal{Q}	propeller torque
$R_{\rm T(Tow)}$	total resistance in towed condition
$R_{\mathrm{T(SP)}}$	total resistance in self-propelled condition
SFC	skin friction correction
Т	propeller thrust
t	thrust deduction factor, e.g. $t = (T - (R_{T(Tow)} - SFC))/T$
$v_{\rm a} = J n_{\rm P} D$	propeller advance speed
w _n	nominal wake, $w_n = \int_0^{2\pi} \int_{d_h/2}^{D/2} ur dr d\theta / (\pi/4) (D^2 - d_h^2)$, where the
	origin is the centre of propeller
$w_{\rm T} = (U_0 - v_{\rm a})/U_0$	Taylor wake fraction
$\eta = (1-t)/(1-w_{\rm T})\eta_0\eta_{\rm R}$	propulsive efficiency
$\eta_0 = JK_{\rm T}/2\pi K_{Q({\rm O})}$	propeller open-water efficiency
$\eta_{\mathrm{R}} = K_{\mathcal{Q}(\mathrm{O})}/K_{\mathcal{Q}}$	relative rotative efficiency

ACKNOWLEDGEMENTS

This work has been partially supported by the U.S. Office of Naval Research under the grants No. N000140210256 (2002–2005) and No. N000140510616 (2005–2008) through Dr P. Purtell and Dr H. Narita. The authors would like to express their appreciation to Mr I. Kuutti and Mr N. Mizutani at NAPA Oy for their valuable discussions and suggestions.

Copyright © 2006 John Wiley & Sons, Ltd.

.

REFERENCES

- 1. Tahara Y, Sugimoto S, Murayama S, Katsui T, Himeno Y. Development of CAD/CFD/Optimizer-Integrated Hull-Form Design System, vol. 240, J. Kansai Society of Naval Architects, 2003; 29–36 (also a related manuscript was presented at NAPA User Meeting 2003, Helsinki, Finland, 2003).
- Tahara Y, Himeno Y, Tsukahara T. An application of computational fluid dynamics to tanker hull form optimization problem. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, 25–27 May 1998; 515–531.
- 3. Tahara Y, Paterson E, Stern F, Himeno Y. Flow- and wave-field optimization of surface combatants using CFD-based optimization methods. 23rd Symposium on Naval Hydrodynamics, Val de Ruil, France, 2000.
- 4. Tahara Y, Ando J, Himeno Y. CFD-based optimization of tanker stern form—minimization of delivered horsepower using self-propulsion simulator. *Practical Design of Ships and Other Floating Structures*, Shanghai, China, 2001, 719–724.
- 5. Tahara Y, Stern F, Himeno Y. CFD-based optimization of a surface combatant. *Journal of Ship Research* 2004; **48**(4):273–287.
- 6. Peri D, Campana EF, Di Mascio A. Development of CFD-based design optimization architecture. *1st MIT Conference on Fluid and Solid Mechanics*. Cambridge, MA, U.S.A., 2001.
- 7. Peri D, Campana EF. High fidelity models in the multi-disciplinary optimization of a frigate ship. 2nd MIT Conference on Fluid and Solid Mechanics. Cambridge, MA, U.S.A., 2003.
- 8. Peri D, Campana EF. Multidisciplinary design optimization of a naval surface combatant. Journal of Ship Research 2003; **41**(1):1–12.
- 9. Peri D, Campana EF. High fidelity models in simulation based design. 8th International Conference on Numerical Ship Hydrodynamics, Busan, South Korea, 2003.
- 10. Peri D, Campana EF. High-fidelity models and multiobjective global optimization algorithms in simulation based design. *Journal of Ship Research* 2005; **49**(3):159–175.
- 11. Campana EF, Peri D, Tahara Y, Stern F. Comparison and validation of CFD based local optimization methods for surface combatant bow. 25th Symposium on Naval Hydrodynamics, vol. 5, St. John's, NL, Canada, 2004; 31–46.
- 12. Tahara Y, Peri D, Campana EF, Stern F. CFD-based multiobjective optimization of a surface combatant. 5th Osaka Colloquium on Advanced Research on Ship Viscous Flow and Hull Form Design by EFD and CFD Approaches, Osaka, Japan, 14–15 March 2005.
- 13. Hino T, Kodama Y, Hirata N. Hydrodynamic shape optimization of ship hull forms using CFD. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka Prefecture University, Japan, 1998.
- 14. Minami Y, Hinatsu M. Multi objective optimization of ship hull form design by response surface methodology. 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan, 2002.
- 15. Newman III JC, Pankajakshan R, Whitfield DL, Taylor LK. Computational design optimization using RANS. 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan, 2002.
- 16. Duvigneau R, Visonneau M, Deng GB. On the role played by turbulence closures in hull shape optimization at model and full scale. *Journal of Marine Science and Technology* 2003; **8**(1):11–25.
- 17. Hoekstra M, Raven HC. A practical approach to constrained hydrodynamic optimization of ships. NAV2003 Conference, Palermo, Italy, June 2003.
- 18. Brizzolara S. Parametric optimization of SWAT-hull forms by a viscous-inviscid free surface method driven by a different evolution algorithm. 25th Symposium on Naval Hydrodynamics, vol. 5, St. John's, NL, Canada, 2004; 47–64.
- 19. Kuutti I. Impacts of product modeling on ship design and production planning. 9th International Conference on Computer Applications in Shipbuilding (ICCAS), Yokohama, Japan, 1997 (also, further information is available at www.napa.fi or NAPA, NAPA Oy Ltd., FIN-00151, Helsinki, Finland).
- 20. Lackenby H. On the systematic geometrical variation of ship forms. Trans INA 92, London, 1950; 289-316.
- 21. Tahara Y, Hayashi G. Flow Analyses Around Downwind-sail System of an IACC Sailing Boat by a Multiblock NS/RaNS Method, vol. 194, J. Society of Naval Architects of Japan, 2003; 1–12.
- 22. Tahara Y, Katsui T, Himeno Y. Development of simulation based design for ship hydrodynamics and fluid engineering. 4th Conference for New Ship & Marine Technology, Shanghai, 2004; 1–13.
- 23. Tahara Y, Wilson R, Carrica P. Comparison of free-surface capturing and tracking approaches in application to modern container ship and prognosis for extension to self-propulsion simulator. *CFD Workshop*, *Tokyo*, 2005, Tokyo, Japan, 9–11 March 2005.
- 24. Nakatake K. A practical method to calculate propulsive performance of ships. *Memoirs of the Faculty of Engineering* 1981; **41**(1):87–122.
- Kijima K, Nakiri Y. Approximate Expression for Hydrodynamic Derivatives of Ship Manoeuvring Motion Taking into Account of the Effect of Stern Shape, vol. 98, J. the West Japan Society of Naval Architects, Japan, 1999; 67–77 [Japanese].

Copyright © 2006 John Wiley & Sons, Ltd.

Int. J. Numer. Meth. Fluids 2006; 52:499–527

- 399 —

- 26. Campana EF, Peri D, Pinto A, Stern F, Tahara Y. A comparison of global optimization methods with application to ship design. 5th Osaka Colloquium on Advanced Research on Ship Viscous Flow and Hull Form Design by EFD and CFD Approaches, Osaka, Japan, 14-15 March 2005.
- 27. Nocedal J, Wright S. Numerical Optimization, Springer Series in Operation Research, Springer: Berlin, 1999.
- 28. Holland J. Adaptation in Natural and Artificial Systems. University of Michigan Press, Ann Arbor, Michigan, 1999
- 29. Davis L. Handbook of Genetic Algorithms. Van Nostrand Reinhold/A Division of Wadsworth, Inc.: New York, 1990.
- 30. Deb K. Multi-Objective Optimization Using Evolutionary Algorithms. Wiley: New York.
- 31. Tahara Y, Katsui T, Kawasaki M, Kodama K, Himeno Y. Development of large-scale high-performance CFD coding method for PC-cluster parallel computing environment-1st Report: Setup and Initial Evaluation of Coding Environment with MPI Protocol, vol. 241, J. Kansai Society of Naval Architects, 2004; 47–58. 32. *Proceedings of the CFD Workshop, Tokyo, 2005*, Tokyo, Japan, 9–11 March 2005.
- 33. Stern F, Wilson RV, Coleman HW, Paterson EG. Comprehensive approach to verification and validation of CFD simulations—Part 1: methodology and procedures. Journal of Fluids Engineering 2001; 123:793-802.
- 34. Wilson RV, Stern F, Coleman HW, Paterson EG. Comprehensive approach to verification and validation of CFD Simulations—Part 2: application for RANS simulation of a cargo/container ship. Journal of Fluids Engineering 2001; 123:803-810.
- 35. Larsson L, Stern F, Bertram V. Summary conclusions and recommendations of the Gothenburg 2000 workshop. Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics, Chalmers University of Technology, September, Gothenburg, Sweden, 2000.
- 36. Baldwin BS, Lomax H. Thin layer approximation and algebraic model for separated turbulent flows. AIAA Paper 78-257, 1978; 1-8.
- 37. SR229 Final Report. Shipbuilding Research Association of Japan, 1999 [Japanese, unpublished].

Copyright © 2006 John Wiley & Sons, Ltd.

ORIGINAL ARTICLE

RANS simulation of a container ship using a single-phase level-set method with overset grids and the prognosis for extension to a self-propulsion simulator

Yusuke Tahara · Robert V. Wilson · Pablo M. Carrica Frederick Stern

Received: December 2, 2005 / Accepted: July 5, 2006 © JASNAOE 2006

Abstract Steady flow simulations for the Korean Research Institute for Ships and Ocean Engineering (KRISO) container ship (KCS) were performed for towing and self-propulsion. The main focus in the present article is on the evaluation of computational fluid dynamics (CFD) as a tool for hull form design along with application of state-of-the-art technology in the flow simulations. Two Reynolds-averaged Navier-Stokes (RANS) equation solvers were employed, namely CFDShip-Iowa version 4 and Flowpack version 2004e, for the towing and self-propulsion cases, respectively. The new features of CFDShip-Iowa version 4 include a single-phase level-set method to model the free surface and an overset gridding capability to increase resolution in the flow and wave fields. The new features of Flowpack version 2004e are related to a self-propulsion scheme in which the RANS solver is coupled with a propeller performance program based on the infinitely bladed propeller theory. The present work is based on a close interaction between IIHR-Hydroscience and Engineering of the University of Iowa and Osaka Prefecture University. In the following article, overviews are given of the present numerical methods and results are presented and discussed for the KCS in towing and selfpropulsion modes, including comparison with available experimental fluid dynamics (EFD) data. Additional

Y. Tahara (🖂)

Osaka Prefecture University, Gakuencho, Sakai 599-8531, Japan e-mail: tahara@marine.osakafu-u.ac.jp

R.V. Wilson

University of Tennessee SimCenter at Chattanooga, Chattanooga, TN, USA

P.M. Carrica · F. Stern

IIHR-Hydroscience and Engineering, Iowa City, IW, USA

evaluation is provided through discussion of the recent CFD Workshop Tokyo 2005, where both methods appeared to yield very promising results.

Key words RANS equations · Container ship · Singlephase level-set method · Self-propulsion simulator

List of symbols

1 1
1
1 1
ьby
R_T
vhere

 R_F is frictional resistance

Deringer

$$\begin{split} C_{p} &= \frac{R_{p}}{\frac{1}{2}\rho U_{0}^{2}S_{0}} & \text{pressure resistance coefficient, where} \\ R_{p} \text{ is pressure resistance} \\ 1+k &= \frac{C_{T}}{C_{F0}} & \text{form factor} \\ C_{F0} & \text{ITTC 1957 frictional coefficient line,} \\ C_{F0} &= \frac{0.75}{(\log_{10}\text{ Re}-2)^{2}} \\ \Delta C_{F} & \text{roughness allowance} \\ D & \text{diameter of propeller} \\ d_{h} & \text{diameter of a propeller hub, } d_{h} = D \times \\ (\text{hub ratio}) \\ J &= \frac{V_{a}}{n_{p}D} & \text{advance ratio} \\ C_{g} & \text{block coefficient} \\ K_{T} &= \frac{T}{n_{p}D} & \text{thrust coefficient} \\ K_{Q} &= \frac{Q}{pn_{p}^{2}D^{3}} & \text{torque coefficient} \\ K_{Q(0)} & \text{torque coefficient} \\ K_{Q(0)} & \text{torque coefficient in open water} \\ (\text{uniform flow}) \\ n_{P} & \text{propeller revolutions [rps]} \\ Q & \text{propeller torque} \\ R_{T(Tow)} & \text{total resistance in towed condition} \\ SFC & \text{skin friction correction} \\ T & \text{propeller thrust} \\ t & \text{thrust deduction factor, e.g.,} \\ t &= \frac{T - \left(R_{T(Tow)} - SFC\right)}{T} \\ v_{u} &= Jn_{F}D & \text{propeller advance speed} \\ w_{u} & \text{nominal wake, } w_{u} &= \frac{\int_{0}^{2\pi} \int_{\frac{\pi}{2}}^{\frac{D}{2}} u dr d\theta}{\frac{\pi}{4} (D^{2} - d_{h}^{2})}, \\ \text{where the origin is the center of} \\ m_{T} & \text{propulsive efficiency} \\ \eta_{0} &= \frac{JK_{T}}{2\pi K_{Q(0)}} & \text{propulsive efficiency} \\ \eta_{k} &= \frac{K_{Q(0)}}{K_{Q}} & \text{relative rotative efficiency} \\ \end{array}$$

D Springer

Introduction

Reynolds-averaged Navier-Stokes (RANS) equation solvers, once developed for evaluating calm water resistance only, are constantly increasing in complexity, and the current generation includes an unsteady capability. In the near future, the same numerical solver will be capable of dealing with resistance, seakeeping, and maneuvering problems. Since 1980, computational fluid dynamics (CFD) workshops on numerical ship hydrodynamics have been organized to assess the status of and define new goals for CFD development.¹⁻³ As the ship hull form was modernized, new challenges for CFD appeared. In the Gothenburg 2000 workshop,⁴ more work on free-surface treatment and inclusion of propeller action effects were suggested, such that the Korean Research Institute for Ships and Ocean Engineering (KRISO) container ship (KCS) was added to one of test cases. Modern container ships have an extended stern overhang, which results in a conditionally wet transom. This geometrical feature yields complexities in the transom wave field, resulting in new CFD challenges. The simulation of self-propulsion is the next goal of CFD so that the method will be a more practical simulation tool to support designers' decision making.

In the present work, towing and self-propulsion simulations were performed for the KCS. The main focus was on the evaluation of CFD as a tool for hull form design, along with application of state-of-the-art technology in the flow simulations. The objectives of the current study were twofold: (1) to demonstrate a free-surface capturing approach with and without overset grid refinement for the KCS towing case to examine relative performance in resolution of the wave field, and (2) to demonstrate a self-propulsion scheme to assess the prognosis for extension of CFD to a self-propulsion simulator. Two RANS equation solvers were employed, namely CFDShip-Iowa version 4 and Flowpack version 2004e, for the towing and the self-propulsion cases, respectively. The new features of CFDShip-Iowa version 4 include a single-phase level-set method to model the free surface and an overset gridding capability to increase the resolution in the flow and wave fields. The new features of Flowpack version 2004e are related to a selfpropulsion scheme, in which the RANS solver is coupled with a propeller-performance program based on the infinitely bladed propeller theory.

The present work is based on a close interaction between IIHR-Hydroscience and Engineering of the University of Iowa and Osaka Prefecture University (OPU) on the development of CFD-based optimization methods for ship design with the long-term objective of including self-propulsion, as investigated herein, in the optimization analysis. In the following article, overviews are given of the numerical methods and results are presented and discussed for towing and self-propulsion models of the KCS, including comparison with available experimental fluid dynamics (EFD) data. Additional evaluation is provided through discussion of the recent CFD Workshop Tokyo 2005,⁵ where both methods were presented.

Summary of the CFD Workshop Tokyo 2005 and current CFD challenges

As already stated, the present results were submitted to the CFD Workshop Tokyo 2005 and detailed evaluations were performed.5 The workshop was held in Tokyo, Japan, on March 9-10, 2005. The purpose of the workshop was to assess the state of the art of CFD for steady and unsteady flows and to accelerate further developments through discussion among the participants, in conjunction with comparisons of computed results from different research groups and with experimental data. Previous related workshops were held in 1980, 1991, 1994, and 2000.¹⁻⁴ Topics discussed at the CFD Workshop Tokyo 2005 were broader compared to the earlier workshops, i.e., predictions of flows and the wave field in the towing condition in still water for KCS and naval combatant vessel DTMB 5415 (fixed, trim-free, and sinkage-free conditions, test cases 1.1-1.3); double model flows for tanker KVLCC2M (test case 1.4); flows and propulsive factors in the self-propelled condition for the KCS (test case 2); flows and hydrodynamic forces and moments in the obliquely towed condition for the KVLCC2M tanker (test case 3); and flows and hydrodynamic forces in the diffraction condition for naval combatant DTMB 5415 (test case 4). In addition, grid dependencies were evaluated by using a common set of computational grids for the KVLCC2M tanker (test case 5). In summary, for steady-state computations, RANS solutions were found to be within acceptable accuracy; however, more effort must be directed toward introduction of anisotropic turbulence models for wake flows, local grid refinement techniques for flows and wave fields, and further developments for unsteady flows associated with computation of motions for resistance and sinkage and trim. In the following article, our focus is on the above-mentioned test cases 1.1 and 2, to which the present results were submitted.

The KCS test case was for the first time selected in the Gothenburg 2000 workshop⁴ for a modern slender ship test case as a replacement for the Series 60 ($C_B = 0.6$) model in the earlier workshop. Figure 1 gives an overview of the KCS hull form. The ship length between



Fig. 1. Overview of Korean Research Institute for Ships and Ocean Engineering (KRISO) container ship (KCS). Ship length between perpendiculars (L_{pp}) is 230 m and 7.2786 m for full and model scales, respectively. The model draft *d* is 0.3418 m, and the model wetted surface area S_0 is 9.4379 m². Towing and self-propulsion experiments were done for a model speed of 2.196 m/s, i.e., $Rn = 1.4 \times 10^7$, Fn = 0.26

perpendiculars (L_{pp}) is 230 m and 7.2786 m for full and model scales, respectively. The model draft d is 0.3418 m and the model wetted surface area S_0 is 9.4379 m². The hull form is characterized by a pronounced bulbous bow, a stern bulb, and an extended stern overhang. The extended stern overhang is commonly seen in recent container ship designs, and it yields some complexities in the stern and wake wave fields. The static waterline shape is normal; however, at operational speeds, the stern wave rises up so that the transom is partially wetted. This creates new numerical and physical modeling challenges for CFD.

EFD studies on the KCS case were performed at the Korean Research Institute for Ships and Ocean Engineering (KRISO, now the Korea Ocean Research and Development Institute) and the Ship Research Institute of Japan (now, the National Maritime Research Institute), where the hull was run with and without an operating propeller for Fn = 0.26 and $Rn = 1.4 \times 10^7$ and for static sinkage and trim conditions.⁶⁻¹¹ The former followed the procedure for ship-point self-propulsion tests with the KP505 propeller to measure propulsive factors. More details on the KP505 propeller are given later. EFD data include resistance, free-surface elevations, mean velocity at several cross sections, hull surface pressure, and propulsive factors associated with propeller open test (POT) data.

The nominal wake at the propeller plane is dominated by after-body inboard-rotating bilge vortices near the center plane. This system is similar to that of tanker hulls, but the magnitude and influence of the vortices are seen to be weaker, e.g., axial velocity contours of the KCS do not exhibit the so-called hook-shaped isovels often exhibited by tankers. The wave pattern is typical for this type of hull for medium Froude numbers: diverging and transverse Kelvin waves in the far field and a steep bow wave and damped stern and wake waves in the near field. The flow features which pose challenges for CFD simulations can be summarized as follows: (A-1) gross aspects of phase and amplitude in the wave profile on the hull and those in the wave cut at y/L = 0.1509, (A-2) the amplitude of the bow wave in the profile, (A-3) shortwave resolution at midship in the wave cut, (A-4) resolution in the global wave field, (A-5) gross features in the transom wave field, (A-6) trends in surface pressure with/without an acting propeller, (A-7) the magnitude and distribution of the mean flow at the propeller section with/without an acting propeller, and (A-8) the accuracy of predicted integral parameters, e.g., hull resistance or propulsive factors.

In test case 1.1, eleven entries from eight countries and nine institutes were presented; nine different CFD codes were used. All methods used a Cartesian coordinate system and solved RANS equations. Only one method used a zonal approach, so that the outer domain was solved as a potential flow. Seven codes used a finite-volume scheme, one used a finite-difference scheme, and one used a finite-analytic scheme. Velocity-pressure coupling algorithms were more varied, i.e., two semiimplicit method for the pressure linked equations (SIMPLE), two pressure implicit with splitting of operators (PISO), four artificial compressibility, and one direct coupling approach. All the finite-volume schemes employed did not use coordinate transformation. Mostly, structured grids were used, including one combination with unstructured grids and two with overset grid methods. Multiblock capability appeared to be a common feature, except for two entries. For the gridding, six of the nine institutes used commercial software, while the others used in-house codes. For turbulence modeling, algebraic models were tested by three entries, oneequation models (Spalart Allmaras) by two entries, and two-equation models by seven entries. As for the twoequation models, the k- ω model was the most popular, and combination with the k- ε model in the outer region and its application to near-wall modification were also considered. One entry used the realizable k- ε model and another tested a two-layer k- ε model. In terms of freesurface treatment, the level-set method was used in five entries, the volume of fluid (VOF) in two entries, and others used a tracking approach.

In test case 2, only four entries from three countries and four institutes were presented, and four different CFD codes were used. As for test case 1.1, all methods used a Cartesian coordinate system and solved RANS equations. Three codes used a finite-volume scheme and one used a finite-analytic scheme in conjunction with one SIMPLE, one PISO, and two other pressure-correction algorithms. Three used multiblock structured grids and one used a single structured grid. Only one institute used an in-house gridding code. Two-equation turbulence modeling was tested by all entries, namely, the $k-\omega$ model, the renormalization group (RNG) $k-\varepsilon$ model, the realizable $k-\varepsilon$ model, and the two-layer $k-\varepsilon$ model. An algebraic model was tested in one entry only. All CFD codes used different approaches for the free-surface treatment, i.e., VOF, level set, tracking, and prescribed pressure distribution methods. For propeller modeling, three CFD codes used a body force approach and one resolved the flow around the actual propeller geometry. In the body force approach, one used a vortex lattice method, one used a prescribed body force approach, and one used the infinitely bladed propeller theory. An actual self-propulsion simulation, where the thrust balances the hull resistance, was demonstrated by two entries.

Most of the above-mentioned methods reproduced many important flow features for both towing and selfpropulsion cases. For the towing case, agreement with EFD data for the boundary-layer and wake flows, the wave field, and the total resistance appears fairly good. For the self-propulsion case, propeller effects are accurately simulated and the resultant propulsive factors show good agreement with EFD data. Indeed, the methods presented in this article yielded the most promising results among all entries, regarding overall quality in flow, wave field, and integral parameters. Therefore, more details of the present method and evaluation of the results will be given.

Uncertainty assessments

In the use of CFD methods, an uncertainty assessment should be provided for the solutions and computational grid. CFD uncertainty assessment consists of verification, validation, and documentation. The simulation uncertainty, U_s , is divided into two components, one from numerical errors, U_{SN} , and the other from the model used, U_{SM} . The approach of Stern et al.¹² is followed here, where modeling and numerical errors were assumed to be completely decoupled, although some level of numerical and modeling error cross-correlation would be expected (e.g., in the near-wall region for turbulence modeling). The U_{SN} value is estimated for both point and integral quantities and is based upon grid and iteration studies, which determine grid (U_{SG}) and iterative (U_{SI}) uncertainties. A root sum square (RSS) approach is used to combine the components and to calculate U_{SN} , i.e., $U_{SN}^2 = U_{SG}^2 + U_{SI}^2$. CFD validation follows the method of Stern et al.¹² and Wilson et al.,¹³ in which a new approach is developed where uncertainties from both the simulation (U_s) and EFD benchmark data (U_D) are considered. Verification formulas were later

Case	RANS code	No. of blocks	Total no. of	No. of	No. of	CPU
			grid poliits	01 03	iterations	nours
Tow-A	CFDShip-Iowa	20	1614232	20	2000	65°
Tow-B	CFDShip-Iowa	22	1934039	22	1000	69 ^d
SP-wop	Flowpack	16 ^a	2 509 760 ^b	1	9000	99 ^e
SP-wp	Flowpack	16 ^a	2 509 760 ^b	1	3000	33 ^e

 Table 1. Computational grids

RANS, Reynolds-averaged Navier-Stokes; SP, self-propulsion; wop, without propeller; wp, with propeller

^a For port and starboard sides

^bFor port and starboard sides

^cOn PC-cluster (CPU: Xeon, 2.4 GHz)

^dOn Origin 3800 (CPU: MIPS R16000)

^eOn Linux-PC (Single CPU: Pentium 4, 2.8 GHz)

Table 2. Turbulence models used for the computations

Remarks	
ar-wall model	
ndard	
ndard	
ndard	

BL, Baldwin-Lomax model; 2L, two layer

^a BL results are presented

corrected in Wilson et al.¹⁴ The first step is to calculate the comparison error, E, which is defined as the difference between the data D (benchmark) and the simulation prediction S, i.e., E = D - S. The validation uncertainty, U_V , is defined as the combination of U_D and the portion of the uncertainties in the CFD simulation that are due to numerical errors U_{SN} and which can be estimated through verification analysis, i.e., $U_V^2 = U_D^2 + U_{SN}^2$. U_V sets the level at which the validation can be achieved. The criterion for validation is that |E| must be less than U_V . Note that for an analytical benchmark, U_D is zero and U_V is equal to U_{SN} . Validation is critical for making improvements and/or comparisons of different models, since U_{SN} is included in U_V .

The present results basically simulate EFD conditions, and both towing and ship-point self-propulsion cases are performed, subsequently referred to as Tow and SP cases, respectively. Table 1 shows CFD cases, RANS codes, and the number of computational grids, whereas Table 2 shows the turbulence models used. Note that there are two CFD results for the Tow case, i.e., Tow-A and Tow-B, which were performed at IIHR and OPU, respectively. Both used the same version of CFDShip-Iowa, but for Tow-A, a non-overset mesh system was used. There are also CFD results for the SP case, i.e., SP-wop and SP-wp, which correspond to the without and with propeller conditions, and these were performed at OPU by using Flowpack. Since the present CFD methods have been evaluated in many precursory studies, and major parts of the codes were taken over

Table 3. Values, uncertainties, and the error for the total resistance

Case	CFD (S)	EFD (D)	U _D (%D)	U _G (%D)	U _I (%D)	U _V (%D)	E (%D)
Tow-A	0.00344	0.00356	1.0	2.0	1.0	2.4	3.4
Tow-B	0.00370	0.00356	1.0	2.0	4.0	4.6	3.8

CFD, computational fluid dynamics; EFD, experimental fluid dynamics; S, simulation results; D, benchmark experimental data; U_D , uncertainty in D; U_G , grid uncertainties; U_L , iteration uncertainty; U_V , validation uncertainty; E, error

from the earlier versions, validation is performed by using estimated grid uncertainties, U_G , from past verification and validation case studies.^{15–18} In addition to the extra resources required for multiple-grid studies, additional complications arise when using overset blocks for multiple-grid studies since the level of overlap on the finest grid must be increased to achieve sufficient overlap on the coarsest grid. The difficulties in using multiple solutions for estimating grid uncertainties should motivate the development of single-grid uncertainty estimation methods. Uncertainties and errors for profile quantities, e.g., wave profiles, are presented as profileaveraged values (based on an L2 norm) in the range of comparison.

Table 3 shows a comparison of uncertainties and values for the total resistance coefficient, $C_{T(Tow)}$. U_D is 1.0%. For Tow-A, U_G and U_I are 2.0% and 1.0%, respectively; and those for Tow-B are 2.0% and 4.0%, respectively. Hence, the resultant U_V values for Tow-A and Tow-B are 2.4% and 4.6%, respectively. Otherwise, |E| is 3.4% and 3.8% for Tow-A and Tow-B, respectively. Since |E| for Tow-B is smaller than U_V , the result is validated for Tow-B at the U_V level.

Table 4 shows a summary of the uncertainties and errors for the wave field results. The profile average ranges are -0.45 < x < 0.45 and -0.45 < x < 1.0 for the wave profile and the longitudinal wave cut, respectively. The resultant U_V for the profile and cut for Tow-A are

Case	U_D	U_G	U_I	$U_{\scriptscriptstyle V}$	Ε
Wave profile	on the hull ^a				
Tow-A	0.92	1	2	2.4	3.7
Tow-B	0.92	1	3	3.3	7.2
Wave cut at	$v/L = 0.1509^{10}$	0			
Tow-A	0.42	5	1	5.1	4.2
Tow-B	0.42	5	3	5.8	4.3
100 2	01.12	0	0	010	

Table 4. Wave profile values and cut errors $(\%\zeta_R)$

^a Profile average for -0.45 < x/L < 0.45, ($\zeta_R = 0.015$)

^b Profile average for -0.45 < x/L < 1.0, ($\zeta_{\rm R} = 0.015$)

Table 5. Uncertainties in the velocity profile ($\% U_0$)

Case	U_D	U_{G}	U_I	U_V	
Tow-A	0.38	2.0	1.0	2.2	
Tow-B	0.38	2.0	2.0	2.8	

Profile average for -0.018 < y/L < 0.018

Table 6. Velocity profile errors ($\% U_0$)

Case	U_D	U_G	U_I	U_V	Еи	Ev	Ew
SP-wop	0.38	5	1	5.1	3.2	2.0	3.1
SP-wp	5	5	1	7.1	5.2	7.2	6.3
	-						

Profile average for -0.018 < y/L < 0.018

2.4% and 5.1%, respectively, and those for Tow-B are 3.3% and 5.8%, respectively. |E| for the profile and cut for Tow-A are 3.7% and 4.2%, respectively, and those for Tow-B are 7.2% and 4.3%, respectively, which yields a conclusion that both CFD results are validated for the cut at the respective U_V levels.

Tables 5 and 6 show a summary of uncertainties and errors for the velocity profiles at the wake section x =0.4911, where a detailed comparison with EFD will later be made. The range of the profile average is -0.018 < y < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.018 < 0.0.018 (z = -0.03). In these tables, results for the Tow and SP cases are included. U_D values for the towing and selfpropulsion cases are $0.38\% U_0$ and $5.0\% U_0$, respectively, where U_0 is the ship speed. The latter is significantly larger than the former, which is apparently due to increased flow complexity caused by the action of the propeller. The resultant U_V for Tow-A and Tow-B are $2.2\% U_0$ and $2.8\% U_0$, respectively. Furthermore, the resultant U_V for SP-wop and SP-wp are $5.1\% U_0$ and $7.1\% U_0$, respectively (Table 6). For this case, the error for each velocity component is also evaluated, i.e, Eu, *Ev*, and *Ew* for the *u*, *v*, and *w* components, respectively. These values for SP-wop are $3.2\% U_0$, $2.0\% U_0$, and $3.1\% U_0$, respectively; and those for SP-wp are $5.2\% U_0$, $7.2\% U_0$, and $6.3\% U_0$, respectively. Especially for the SPwp case, the errors for the cross-flow components are larger than those for the axial-velocity component,

which is due to the more drastic influences of the rotating propeller for the former. It is shown that for SP-wop, the errors are smaller than U_V , hence the results are validated at the U_V level. For SP-wp, errors for u and w are smaller than U_V , so that the results are validated at the U_V level. The result for w may be judged nearly validated, since the error is almost equivalent to the U_V value.

Towing case simulation

RANS equation solver: CFDShip-Iowa version 4

CFDShip-Iowa is a general-purpose, multiblock, highperformance parallel computing, unsteady RANS code developed for computational ship hydrodynamics. The RANS equations are solved using higher-order upwind finite differences, PISO, and an isotropic blended $k-\omega/k-\varepsilon$ two-equation turbulence model. The free surface is modeled using a single-phase level-set approach. Details related to the governing equations and numerical methods can be found in Wilson et al.^{19,20} and in Carrica et al.^{21,22}

The earlier version of CFDShip-Iowa (version 3), which used the free-surface tracking approach, was successfully applied to predict the resistance and the wave and flow fields around the surface combatant vessel DTMB 5415 at the previous workshop.¹⁵ In addition, this approach has been applied to the forward speed diffraction problem, where ships encounter regular head waves and advance with constant speed;²³ prediction of roll decay motion for a surface combatant;²⁴ and pitch and heave motions for the Wigley hull.²⁵ However, the surface tracking approach is limited to flows with small to medium wave slopes and cannot resolve steep, overturning, or breaking waves due to difficulties in the grid conforming process.

In an effort to remove these limitations, a steady and unsteady single-phase level-set method was developed for CFDShip-Iowa to handle both complex ship geometry and complex interfacial topology due to higher Froude numbers, bluff geometry, and/or largeamplitude motions and maneuvering. Since discussing in detail the single-phase method would require significant space, the authors refer readers to Wilson et al.^{19,20} and Carrica et al.^{21,22} Overset grids are used to provide flexibility in grid generation, local grid refinement, and for bodies and/or blocks with relative motions. Subsequently, the single-phase level-set approach was applied to the prediction of breaking waves at higher Froude numbers¹⁹ and for the forward speed diffraction problem,²¹ among other applications.

CAD-interfaced automatic grid generator

At present, CAD methods have essentially replaced the drawing board as the basic tool for hull form design. In the use of a CFD tool, a major task will be the simulation setup, including gridding, which is still regarded to be a serious bottleneck by designers. In the present work, a recently developed CAD-interfaced automatic grid generator was applied. The method was originally developed for a CFD-based hull form optimization scheme.²⁶

Surface, as well as volume, grids are automatically generated, based on prescribed setup parameters (which are basically the same as those in series case studies if the grid topology is fixed). The volume grid is generated by an elliptic algebraic method using an exponential scheme and the method of lines. A concern for automatic gridding is the robustness for practical complexities at the hull surface. Through preliminarily exercises, the present scheme was shown to be capable of application to tankers, surface combatants, and container ships. In all cases, the grid orthogonality, especially near the hull surface, was sufficiently maintained. This gridding scheme is demonstrated for the Tow-A case.

Computational grid, conditions, and environment

For the present condition, two simulations were performed, one each by IIHR and OPU, by using the same RANS code, CFDShip-Iowa version 4, but with different gridding strategies. These are described below in association with the computational environment. Both simulations basically simulate EFD conditions, i.e., Fn =0.26 and $Rn = 1.4 \times 10^7$, and static sinkage and trim conditions. In addition, Tables 1 and 2 show the number of computational grids, CPUs used for parallel computing, iterations, and computational hours to achieve sufficient convergence; the turbulence models used in the present simulations are also indicated. The conditions are further described below.

Tow-A

Figure 2 gives an overview of the computational grid for the case Tow-A. The grid was generated by using the previously mentioned automatic gridding scheme. An overset grid was not used for this case and only the starboard side was included in the solution domain. The topology was a C-O type, and the domain covered one ship length upstream of the bow, out from the side and bottom of the hull, and downstream of the stern. A noslip boundary condition was enforced at the hull surface, where the first grid point away from the body surface was located at around $y^+ = 1$. The domain was decom-



Fig. 2. Computational grid for the towed condition, Tow-A. The grid was generated by using an automatic gridding scheme. An overset grid was not used for this case. The starboard side only was included in the solution domain

posed into 20 blocks. The total number of grid points was 1614232, in which the distribution was nearly equivalent to that of the background grid for Tow-B. Through comparison of the Tow-A and Tow-B results, the effectiveness of the overset grid scheme was evaluated.

The computation was performed on a 64-CPU PCcluster system (CPU: Xeon, 2.4 GHz). The number of CPUs is the same as the number of blocks. In total, 2000 global iterations were required to achieve sufficient convergence and that took 65 h. In the present simulation, a blended k- ω/k - ε model was used with a near-wall (low Reynolds number) model, which finally appeared to be the best option (at present, the available options are: the original blended k- ω/k - ε model; the blended model with the share-stress transport (SST) model; and the blended model with a near-wall model).

Tow-B

In contrast to Tow-A, for Tow-B a state-of-the-art overset gridding technique was used. The KCS solution domain and grid are shown in Figs. 3 and 4. The solution domain extends one ship length upstream and downstream of the ship. Figures 3a, b show the solution domain, including background and overset refinement blocks for the free surface and nominal wake plane. An overset block was used near the free surface to improve the resolution of the Kelvin wave system (Fig. 3c).



Fig. 3. Solution domain with a overset free-surface refinement blocks, **b** nominal wake and free-surface refinement blocks (under free-surface perspective), **c** free-surface refinement grid (*top*) and Kelvin wave system (*bottom*) and, **d** grid and solution details at the nominal wake plane, x/L = 0.4911

Figure 3d shows details of the nominal wake refinement block, which was used to improve the resolution of the bilge vortices.

The base background grid (before decomposition for parallel processing) consisted of a hyperbolically generated near-hull grid and an elliptically smoothed far-field grid, both with an O-O topology. The near-hull grid extended from the hull surface to roughly 0.03*L* away



Fig. 4. Center-plane grid details at the bow (*top*) and stern with nominal wake refinement blocks (*bottom*)

from the hull, while the far-field grid extended from 0.03L to the far-field boundary located at one ship length. Details of the near-hull and far-field grids at the bow and stern center plane are shown in Fig. 4.

A no-slip boundary condition was enforced at the hull surface, where the near-wall spacing was set to yield $y^+ < 1$ for turbulence modeling considerations. The total number of grid points was 1934039. The base background, nominal wake refinement, and free-surface refinement blocks were decomposed for parallel processing, which yielded a 22-block grid system. The computational grid was generated using the commercial software Gridgen (Pointwise, Fort Worth, TX, USA), which creates non-overlapping patched multiblock boundaries. Two rows of ghost cells were dynamically added by the CFDShip-Iowa code at run time to all multiblock boundaries to ensure enforcement of the flow field equations and to maintain higher-order accuracy.

The simulation took roughly 69 wall clock hours to run 1000 iterations using 22 processors of an Origin 3800 machine. In the present simulation, a standard blended $k-\omega/k-\varepsilon$ model was used.

Results

In the following sections, the computational results for the towing case are discussed. We consider a ship fixed in a uniform onset flow using a Cartesian coordinate system with the origin on the undisturbed free surface at the midship, the x and y axes on the horizontal plane (x is in the longitudinal direction), and the z axis directed vertically upward. In the presentation of results, the length and velocity scales are nondimensionalized using the ship length (L_{nn}) and ship speed (U_0) , respectively; the pressure is nondimensionalized using the stagnation pressure $0.5\rho U_0^2$, where ρ is the fluid density. The hydrodynamic force of main interest, i.e., the total resistance in the present study, is expressed in a coefficient form as $C_T = R/0.5\rho S_0 U_0^2$, where S_0 is the wetted surface area of the ship for the static upright condition. As noted earlier, x = -0.5 and 0.5 correspond to the forward perpendicular (FP) and after perpendicular (AP), respectively.

Resistance

First, the total resistance, C_T , is discussed. Table 3 shows a comparison of C_T between EFD and the present results. EFD- C_T is 0.00356, in contrast to 0.00344 and 0.00370 for Tow-A and Tow-B, i.e., the comparison errors E are 3.4%EFD- C_T and 3.8%EFD- C_T , respectively. The C_T can be subdivided into pressure and frictional parts, and for Tow-A these are 21%EFD- C_T and 75.6%EFD- C_T , respectively; for Tow-B they are 22.5%EFD- C_T and 81.5%EFD- C_T , respectively. Note that the corresponding ITTC-57 and Schoenherr values for frictional resistance are 0.0028 and 0.00278, respectively, whereas the predicted values by the present results are 0.0027 and 0.0029 for Tow-A and Tow-B, respectively.

The agreement between the present CFD and EFD results is considered to be acceptable for this level of grid size, since it was reported in the Gothenburg 2000 Work-shop⁴ that the average value for all participants' for this case was about 5.2% higher than EFD- C_T , and the variation was 4.5% EFD- C_T . Interestingly, the CFD results presented at the Tokyo 2005 Workshop⁵ were more diverse, i.e., the averaged value of all participants' was about 11.2% higher than EFD- C_T , and the variation was 15% EFD- C_T . This variation was due more to the pressure than frictional components, i.e., the averaged value and variation for the former are 20.9% EFD- C_T and 7.9% EFD- C_T and 4.8% EFD- C_T , respectively.

Wave field

Next, the results for the wave fields are discussed. Figure 5 shows the global wave contours for the two numerical results, i.e., Tow-A and Tow-B, and EFD performed at KRISO. In general, the predictions show good agreement with experimental values in terms of both the magnitude and location of the peaks and troughs of the Kelvin pattern. In fact, the agreement between Tow-B and EFD is excellent, and the numerical dissipation of the Kelvin wave into the far field is reduced by the use of an overset refinement block (Fig. 3c). More details in wave features, including short wave systems, were captured. Simulation without the free-surface refinement grid (i.e., Tow-A) displays noticeable dissipation of the Kelvin wave leading to larger differences with the EFD data. For both results, comparison of the bow wave field shows good agreement with the experiments; however, the wave field in the vicinity of the transom corner appears to be somewhat under-resolved, and could probably be improved with the addition of an overset refinement block.

Figure 6 shows closer views of the transom wave field. Similar features of the wave field were predicted, i.e., the particular aspects often seen around partly wetted transoms, e.g., the characteristic horizontal V-type extent of the damped wave region. However, more detailed evaluation of the results is unfortunately precluded by the



Fig. 5. Comparison of wave contours. *Exp*, experimental results; *A*, computational fluid dynamics (CFD) Tow-A results; and *B*, CFD Tow-B results



Fig. 6. Wave contours near the transom stern. *A* and *B* correspond to CFD Tow-A and Tow-B, respectively

limited resolution of EFD data. It is noteworthy that the present strategy of grid arrangement near the transom, i.e., a local C-type topology used for both results, is capable of adequately representing the transom geometry. A related issue was often reported at the workshops: not many CFD simulations predict the wave field if the transom geometry is not correctly represented.

Figures 7 and 8 show wave profiles at the hull and at a longitudinal wave cut (y = 0.1509), and enable more critical evaluation of the present CFD results. In these figures, values of U_V and E are also shown; the respective U_{ν} for the wave profile and the cut are seen to be similar. Both results successfully capture the gross features of the EFD data, and E for both results is mostly within the estimated U_V range. It is noteworthy that the differences between the two CFD results are likely within the combined U_{SN} range, i.e., $(U_{SN(T_{OW}-A)}^2 + U_{SN(T_{OW}-B)}^2)^{0.5}$, which is about 3.8% and 7.7% for the wave profile and the cut, respectively. On the other hand, some shortcomings can be seen: an underpredicted wave-cut amplitude near the end of the plot region, a missed wave-cut short-wave system around midship, and an underand overpredicted profile wave elevation at the bow-wave crest and near the stern. An underprediction of the bowwave crest was a general trend in the workshop results,



Fig. 7. Wave profile on the hull (*top*) and errors and validation uncertainties (*middle* and *bottom*). *Circles* and *lines* represent experimental fluid dynamics (EFD) and CFD data, respectively. *A* and *B* correspond to CFD Tow-A and Tow-B, respectively



Fig. 8. Comparison of longitudinal wave cut at y = 0.1509 (*top*) and errors and validation uncertainties (*middle* and *bottom*). *Circles* and *lines* represent EFD and CFD data, respectively. *A* and *B* correspond to CFD Tow-A and Tow-B, respectively

except for a few of results, namely one that used freesurface tracking and two that used free-surface capturing approaches. Wave damping occurring in the very far field and downstream is noticeable, although that for Tow-B is less significant. In general, the wave damping correlates with the transverse grid resolution.

Boundary layer and wake flow and surface pressure field

Figures 9 and 10 show the axial velocity contours and cross-plane vectors for several representative cross sections, i.e., x = 0.4, 0.45, and 0.4825, for CFD (Tow-B) results and EFD data. Figure 10 also shows the axial vorticity contours. It can be seen that the rapid contraction of the afterbody from x = 0.4 to 0.45 results in a dramatic thickening of the boundary layer and clockwise (shown in blue, Fig 10b) and counterclockwise (shown in red, Fig. 10b) bilge vortices. At x = 0.45, the afterbody begins the transition to the propeller hub, resulting in the formation of a third, smaller clockwise vortex just downstream of the termination of the propeller hub (shown in blue, Figs. 10c and 10d).

The measurements show some evidence of the three counter-rotating vortices (Fig. 10d), although the measurements are not present at x = 0.4825 and the measurement grid was too coarse at x = 0.4911 to verify the existence of the clockwise vortex downstream of the propeller hub. Overall, the agreement of the axial velocity contours in Fig. 9 is acceptable for the resolution of the present grid. The largest differences appear to be in the shape of the higher speed contours (i.e., U = 0.8-0.9). The experimental shape has a more pronounced bulge and a smaller boundary layer thickness (e.g., y, z = -0.03, -0.02, Fig. 10a) compared to the CFD predictions. Use of the overset refinement block at the nominal wake plane was required to resolve the small-scale details of the bilge vortices.

Figure 11 shows the wake flow near the propeller section, i.e., x = 0.4911, for Tow-A and Tow-B, and Fig. 12 shows the EFD results for the same section. The section is actually just after the propeller center section at x = 0.4825, and was selected by the aforementioned CFD workshops to facilitate comparison between the cases without and with an acting propeller. The latter is the focus of the self-propulsion condition, which will be discussed later. Both results capture general aspects of the flow shown in the EFD data, e.g., the distribution of the wake, the minimum value of the axial velocity, and the location of vortices and associated redistribution of the wake. The plateau-like area around z = -0.03, which is associated with the vortex centered at z = -0.025, is fairly well reproduced. Figure 13 shows the CFD and EFD velocity profiles at z = -0.03 at the same section and

enables more detailed and critical examination of the present results. The agreement with the data is good; however, both results are seen to miss details in the distribution of the transverse velocity component (v) near the center plane. The wake predicted by Tow-B is somewhat narrower than that by Tow-A, and the differences are likely attributed to differences in the turbulence model. As shown for Tow-A, inclusion of near-wall effects in the k- ω equation appears to be effective.

Figure 14 shows the EFD and CFD results for the surface pressure field near the stern. The overall features shown in the two results are similar and agree well with those shown in the EFD data, e.g., the location of the zero contour is nearly identical. A characteristic low-pressure region near the stern and bilge is predicted by both results. In fact, this success is expected based on the above discussions of the resistance and boundary layer and wake flow, since they are closely related to the surface pressure field associated with the generation of three-dimensional flow separation that occurs in the stern region. Some differences are seen between CFD and EFD for contours near the bilge in the negative value region, i.e., more complexities, including isolated islands, are shown in the EFD results.

Self-propulsion case simulation

RANS equation solver: Flowpack version 2004e

Flowpack version 2004e²⁷ was used for simulation of the self-propulsion case. The code was developed particularly for CFD education and research, and design applications for ship hydrodynamics, aerodynamics, and fluid engineering. For design applications, multiblock domain decomposition and propeller–hull interaction are included. At present, Flowpack has a tight interface with both commercial and the authors' in-house grid generators. A summary of applications is available in Tahara et al.²⁸ for full-scale simulation, hull form optimization, the America's Cup downwind sail system, a fully appended sailing boat, parachutes, automobiles, and others.

The numerical method of Flowpack solves the unsteady RANS equations and continuity equations with a zero or two-equation turbulence model for the mean velocity, pressure, and eddy viscosity or turbulence parameters by using a body/free-surface conforming grid. The equations are transformed from Cartesian coordinates in the physical domain to numerically generated, boundary-fitted, nonorthogonal, curvilinear coordinates in the computational domain. A partial transformation is used, where the coordinates are transformed but the



Fig. 9a–d. Axial velocity contours from experiment (*left*) and CFD (*right*) at x/L = 0.40 (a), 0.45 (b), 0.4825(c), and details at x/L = 0.4825 (d)



Fig. 10a–d. Cross-flow vectors from experiment (*left*) and CFD with axial vorticity (*right*) at x/L = 0.40 (**a**), 0.45 (**b**), 0.4825 (**c**), and details at x/L = 0.4825 (**d**)



Fig. 11. Comparison of axial-velocity contours (*left*) and crossplane vectors (*right*) at x = 0.4911 (just after the propeller center section, x = 0.4825) for the towed condition (without propeller). *A* and *B* correspond to CFD Tow-A and Tow-B, respectively

velocity components are not. The equations are solved using a regular grid, finite-analytic spatial and first-order backward-difference temporal discretization, a PISOtype pressure algorithm, and the method of lines.

Flowpack uses a free-surface tracking approach to resolve wave fields. Exact nonlinear kinematic and approximated dynamic free-surface conditions are applied on the free surface, which is determined as part of the solution; i.e., the dynamic conditions are applied to velocity and pressure, and the free-surface elevation is determined through the solution of the nonlinear kinematic condition using a Beam and Warming linear multistep scheme with both explicit and implicit fourthorder artificial dissipation. The computational grid is automatically updated at each time step to conform to both the body and the free surface.

Self-propulsion scheme

The present self-propulsion scheme was originally developed for application to a CFD-based tanker stern opti-



Fig. 12. Measured axial-velocity contours (*top*) and cross-plane vectors (*bottom*) for the towed condition (without propeller). The location is just after the propeller section, i.e., x = 0.4911. Other conditions coincide with those of the simulation, i.e., $Rn = 1.4 \times 10^7$, Fn = 0.26

mization for minimum delivered power.¹⁸ Figure 15 shows the overall computational procedure of the present self-propulsion scheme. The method consists of three parts, i.e., a RANS solver, a propeller performance program, and a root finder module to determine the propeller rotational speed n_p so that the hull part of the resistance *R* balances with the propeller thrust *T*:

$$T = R \tag{1}$$

In application for the ship-point test, the skin-friction correction, i.e., *SFC*, must be included so that:

$$R = R_{T(SP)} - SFC \tag{2}$$

where $R_{T(SP)}$ is the hull resistance for the self-propulsion state. Note that *SFC* for the present KCS test case is given from EFD by:

$$SFC = \frac{\rho U^2 S_0}{2} \left\{ (1+k) (C_{F0M} - C_{F0S}) - \Delta C_F \right\}$$
(3)

Deringer



Fig. 13. Comparison of velocity profiles at x = 0.4911 and z = -0.03 for the towed condition (without propeller). *Circles* and *lines* represent EFD and CFD data, respectively. *A* and *B* correspond to CFD Tow-A and Tow-B, respectively



Fig. 14. Comparison of surface pressure contours near the stern $(C_P \text{ contours})$ for the towed condition (without propeller). *A* and *B* correspond to CFD Tow-A and Tow-B, respectively

$$\begin{cases} C_{F0M} = 2.832 \times 10^{-3} \text{ for } R_{nM} = 1.4 \times 10^{7} \\ C_{F0S} = 1.378 \times 10^{-3} \text{ for } R_{nS} = 2.39 \times 10^{7} \\ \Delta C_{F} = 0.27 \times 10^{-3} \end{cases}$$
(4)

Deringer



Fig. 15. Block diagram of the present self-propulsion scheme. The root-finder module determines the propeller rotational speed n_p so that the hull resistance *R* balances the propeller thrust *T*

Subscripts M and S correspond to model and full scales, respectively. Propulsion factors are evaluated when the self-propulsion state is achieved. For example, the thrust deduction factor is given by:

$$1 - t = \frac{R_{T(T_{ow})} - SFC}{T} = \frac{R_{T(T_{ow})} - SFC}{R_{T(SP)} - SFC}$$
(5)

where $R_{T(Tow)}$ is the hull resistance for the towing state.

Propeller action effects are included in the RANS equations by a body force approach. The body force distribution is interactively and iteratively determined by propeller performance calculation based on the infinitely bladed propeller theory²⁹ by using a free vortex distribution in the wake and a bound vortex distribution on the propeller disk. The disk is divided into $36 \times 5 = 180$ panels in the present work.

Computational grid and conditions

A ship-point self-propulsion simulation was performed at OPU by using the above-mentioned RANS code and



Fig. 16. Computational grid for the self-propulsion case. Both port and starboard sides are included to simulate asymmetric flows caused by the action of the propeller

self-propulsion scheme. The simulation conditions basically follow those of EFD, i.e., Fn = 0.26, $Rn = 1.4 \times$ 10^7 , and static sinkage and trim conditions. The already discussed automatic gridding scheme was used to generate the computational grid. Figure 16 shows an overview of the grid. Both port and starboard sides are included in the solution domain to simulate the asymmetric flows generated by action of the propeller. The grid topology is an O-O type, and the total number of grid points is 2509760. The domain covers one ship length upstream of the bow, one ship length out from the side and bottom of the hull, and one ship length downstream of the stern. The first grid points away from the body surface are located around $v^+ = 1$.

As already stated, the infinitely bladed propeller theory was used in the present simulation, in which the propeller is represented as a circular disk. Figure 17 shows the location of the propeller disk. The center of the disk coincides with that of the real propeller model used in EFD, i.e., (x, y, z) = (0.4825, 0, -0.02913). Figure 18 shows the KP505 propeller model used for both CFD and EFD. The propeller has five blades, an NACA66 section profile, and a hub ratio of 0.18. The propeller diameter is 7.9m and 250mm for full and model scales, respectively. Figure 19 shows EFD data from a propeller open test (POT) carried out at the National Maritime Research Institute (NMRI) for the CFD Workshop Tokyo 2005. The POT data are used for propeller modeling in the infinitely bladed propeller theory.



Fig. 17. Location of the propeller disk in the present selfpropulsion simulation. The propeller center is located at (x, y, z) =(0.4825, 0, -0.02913)



Fig. 18. The propeller model used for the self-propulsion simulation and experiment is named KP505 and was prepared for the Gothenburg 2000 and Tokyo 2005 workshops. It has five blades with a NACA66 section profile and a hub ratio of 0.180. The propeller diameters are 7.9m and 250mm for full and model scales, respectively

The self-propulsion simulation was performed by using the towed condition results as an initial guess. Technically, these are with- and without-propeller conditions, and hereafter, these computations are referred to as SPwp (with propeller) and SP-wop (without propeller), respectively. The number of iterations and wall clock hours are shown in Table 1. SP-wop was performed on a single CPU PC (VT-WSi, CPU: Pentium 4, 2.8 GHz). In fact, the influence of the propeller is limited to the flow in the stern and wake regions. Therefore, solution convergence is achieved in a relatively small number of iterations: 9000 and 3000 global iterations for SP-wop and SP-wp, respectively, were found to yield sufficient convergence of solutions. The wall clock hours for these computations were 99 and 33h, respectively.



Fig. 19. Experimental data from the propeller open test (POT) for the KP505 propeller. The experiments were carried out at the National Maritime Research Institute for the Tokyo Workshop 2005. These data were used in the propeller performance calculation implemented in the present self-propulsion scheme

Table 2 shows the turbulence models used in the present CFD cases. The Baldwin-Lomax model (BL: original form) and two-layer $k \cdot \varepsilon$ model (2L $k \cdot \varepsilon$) were used and the results were examined. It was found that the 2L k- ε results were slightly better for cross-flow prediction at the propeller section. However, its use does not indicate a clear improvement in the axial velocity. The total resistance for the 2L k- ε model was about 2% lower than the EFD data, and that is mainly attributed to an underestimated frictional resistance. Another drawback for the present application of the 2L k- ε model is the larger iteration uncertainty U_I than for the BL results. Better performance for the BL model was somewhat unexpected, and that may correlate with the grid resolution problem near the wall, since the 2L k- ε model generally needs more grid points near the wall. Finally, further attempts to improve the 2L k- ε results were not pursued. For a focus on the application to practical design, the results obtained by simpler turbulence models are still considered to be valuable, and show the capability of such a model for highly complex flow simulations.

Results

In the following sections, computational results are discussed for the self-propulsion simulation. Definition of the coordinate system and the nondimensional presentation of results are the same as those for the towing

Table 7. Comparison of propulsive factors

Factor	CFD	EFD
$C_{T(T_{out})}$	0.003545	0.003550
$1-w_n$	0.634	0.686
$C_{T(SP)}$	0.00393	_
K_T	0.1670	0.1700
K _o	0.0282	0.0288
1-t	0.8515	0.8530
$1-w_{\tau}$	0.789	0.792
n	0.631	0.682
n_R	1.074	1.011
n	0.732	0.740
J	0.718	0.728
n_P (rps)	9.528	9.500

simulations. Other definitions of propulsive factors are shown in the List of symbols.

Propulsive factors

As already stated, the SP-wp case was performed by using the SP-wop results as the initial guess. When the self-propulsion state was achieved, i.e., the propeller thrust balanced the hull resistance minus *SFC*, all propulsive factors were directly evaluated. Table 7 shows the propulsive factors for the CFD and EFD results, and generally good agreement is demonstrated. Due to the action of the propeller, the hull resistance increases about 10%, which is the general trend shown in the results presented at past workshops.^{4,5} The increase in hull resistance is mainly due to an increase, but the magnitude is insignificant.

Wake flows

Figure 20 shows the calculated axial-velocity contours and cross-plane vectors at x = 0.4911, which is just after the propeller center section, i.e., x = 0.4825. Figure 21 shows the measured axial-velocity contours and crossplane vectors for the self-propulsion condition, and through comparison with Fig. 12, salient flow features resulting from the action of the propeller can be examined. Furthermore, Fig. 22 shows the velocity profiles for CFD and EFD at x = 0.4911 and z = -0.03, which enables a more critical evaluation of results. CFD results for the without-propeller condition are qualitatively nearly identical to those for the Tow-A and Tow-B cases: i.e., CFD captures the general aspects of the flow shown in the EFD data, e.g., the distribution of the wake, the minimum value of the axial velocity, and the location of vortices and associated redistribution of the wake. However, underprediction of the secondary flow magnitude is



Fig. 20. Axial-velocity contours and cross-plane vectors at x = 0.4911 (just after the propeller center section, x = 0.4825). *Top*, without propeller; *middle* and *bottom* with propeller

more significant, which is likely due to the limitations of the simple turbulence model.

As shown in the EFD data, the influences of propeller action on the flow are drastic, especially in the region right after the propeller. The flow exhibits characteristics of an asymmetric swirling jet and accelerated velocity fields, and importantly, larger acceleration of flow on the starboard side than on the port side for the given propeller rotational direction. The present CFD model successfully captures such general features of the flow. In a more critical evaluation, the cross-flow velocity components again indicate some discrepancies from the EFD data, especially for the transverse component near the hub center. In part, the lack of hub effects in



Fig. 21. Measured axial-velocity contours (*top*) and cross-plane vectors (*bottom*) for the self-propulsion condition (with propeller). The location is just after the propeller section, i.e., x = 0.4911. The other conditions coincide with those of the simulation, i.e., $Rn = 1.4 \times 10^7$, Fn = 0.26

the present propeller model may be responsible for this discrepancy.

Figure 23 shows the effective wake, propeller-induced axial velocity, and axial body force contours on the propeller disk. Note that the effective wake u_e is defined as $u_e = u_t - u_p$, where u_t and u_p are the total and propeller-induced axial velocity, respectively. Due to the propeller-hull interaction, an asymmetric distribution of these values is evident. The axial body force directly correlates with u_p and the flow just downstream of the propeller, where the flow is dominated by the propeller-induced effects.

Surface pressure field

Figures 24 and 25 show surface pressure contours for the SP-wop and SP-wp cases. Figure 24 shows the side view and includes EFD data; Fig. 25 shows the back view. Comparison with EFD data for the towing condition in Fig. 14 shows the specific influences caused by the action of the propeller. The SP-wp results clearly show expected



Fig. 22. Velocity profiles at x = 0.4911 and z = -0.03. *Top*, without propeller; *bottom* with propeller. *Symbols* and *lines* represent EFD and CFD data, respectively

pressure decreases due to the action of the propeller, which is more evident in the region where the boundary layer is thick, i.e., the region near the stern bulb and near the propeller. The trends are also shown in the EFD data as a backward shift of pressure contours, which agrees well with that predicted by CFD. It is also shown that the influence of the propeller is restricted to the stern region, as is commonly assumed in hull form design. As shown in Fig. 25, the CFD results indicate a somewhat asymmetric distribution of surface pressure. This is considered to be due to the propeller–hull interaction, in which the body force, as well as the propeller-induced velocity, is asymmetric on the propeller disk.

Conclusion and prognosis

KCS towing and self-propulsion CFD simulations were performed. Two RANS equation solvers were employed, namely CFDShip-Iowa version 4 and Flowpack version 2004e, for the towing and self-propulsion cases, respectively. The present work was based on a close interaction between IIHR-Hydroscience and Engineering of the University of Iowa and Osaka Prefecture **Fig. 23.** Effective wake, propeller-induced axial velocity, and axial body force contours on the propeller disk for the with-propeller condition ($Rn = 1.4 \times 10^7$, Fn = 0.26): **a** effective wake, **b** propeller-induced axial velocity, and **c** axial body force





Fig. 24. Surface pressure contours near the stern (C_P contours). *Cal*, calculated values; *prop*, propeller



Fig. 25. Surface pressure (C_P) contours without and with a propeller ($Rn = 1.4 \times 10^7$, Fn = 0.26). The *circle* indicates the location of the propeller

University. In the present article, an overview of the numerical methods is given and results are presented and discussed for the KCS towing and self-propulsion cases, including comparison with available EFD data. Additional evaluation is provided through discussion of the recent CFD Workshop Tokyo 2005, where both methods were presented. Indeed, the methods presented in this article yielded the most promising results among all entries, regarding overall quality in flow, wave field, and integral parameters.

The free-surface capturing approach based on the single-phase level-set method is shown to be very promising. The gridding for the approach is easier since special care for wave conforming is no longer needed. Also, numerical wave damping in the outer region is shown to be smaller, and more short-wave systems are efficiently captured. The correct vertical and horizontal grid distribution is still a point of concern, but this will be overcome by parametric and automatic gridding techniques. CAD-interfaced automatic gridding schemes such as that demonstrated in the present work will help alleviate the burden of grid generation for the designer.

Results based on a propeller model demonstrated in the present work indicate practical accuracy in design. Therefore, the method is recommended for design applications at the present level. However, investigation of more advanced models such as a rotating propeller model must be continued, since that will solve the intrinsic problem of the simpler potential flow-based model, e.g., treatment of free vortices and viscous correction. The much heavier computational load may be a drawback, but this can be overcome by the use of highperformance computing.

In summary, the current status of CFD demonstrated in the present work is such that the accuracy of the methods will be acceptable in practical design, after performing extra series-hull case studies to guarantee the trends in the current solutions. Furthermore, the present automatic grid generator, appropriate free-surface modeling, and self-propulsion scheme can be integrated to develop a self-propulsion simulator by introducing several additional components. These are an overset grid approach for more accuracy of flow and wave fields in the regions of interest, a sinkage and trim prediction scheme, and consideration of appendages such as struts or rudders.

In addition, the present results and those from other test cases show the ability of the method to accurately and efficiently resolve the flow and wave fields around practical surface ships. The method is currently being applied to ship research in the areas of resistance and propulsion, seakeeping, and maneuvering. Future work includes extensions for automation of the grid refinement process and use of dynamic overset grids for largeamplitude ship motions and maneuvering and for bodies or blocks with relative motions. Other efforts include application of detached eddy simulation and development of robust and efficient nonisotropic turbulence models for practical surface ships.

Acknowledgments. This research was supported by the Office of Naval Research under grants N00014-05-1-0616 and N00014-02-1-0304 under the administration of Dr. Patrick Purtell. The authors would like to acknowledge the DoD High Performance Computing Modernization Office. Some simulations were performed at the Aeronautical Systems Center Major Shared Resource Center using the Origin 3800 machine.

References

- Larsson L (ed) (1981) SSPA-ITTC workshop on ship boundary layers. SSPA report, no.90. SSPA Publication, Gothenburg, Sweden
- 2. Larsson L, Patel VC, Dyne G (eds) (1991) Flowtech research report, no.2. Flowtech International, Gothenburg, Sweden
- Kodama Y (ed) (1994) CFD Workshop Tokyo. Ship Research Institute, Ministry of Transport and Ship and Ocean Foundation, March 22–24, Tokyo, Japan, vol 1 and 2
- Larsson L, Stern F, Bertram V (2000) Summary conclusions and recommendations of the Gothenburg 2000 workshop. Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics. Chalmers University of Technology, Gothenburg, Sweden

- Hino T (ed) (2005) CFD Workshop Tokyo 2005. National Maritime Research Institute, March 9–11, Tokyo, Japan
- Van SH, Kim WJ, Kim DH, et al (1997) Measurement of flows around a 3600TEU container ship model (in Korean). Annual Autumn Meeting, SNAK, Seoul, Korea, pp 300–304
- Lee J, Lee SJ, Van SH (1998) Wind tunnel test on a double deck shaped ship model. In: The Third International Conference on Hydrodynamics, Seoul, Korea, pp 815–820
- Van SH, Kim WJ, Yim GT, et al (1998) Experimental investigation of the flow characteristics around practical hull forms. In: Third Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, pp 215– 227
- Fujisawa J, Ukon Y, Kume K, et al (2000) Local velocity field measurements around the KCS model (SRI M.S.No.631) in the SRI 400-m towing tank. Ship Performance Division Report No. 00-003-02, The Ship Research Institute of Japan, Mitaka, Tokyo
- Tsukada Y, Hori T, Ukon Y, et al (2000) Surface pressure measurements on the KCS Model (SRI M.S.No.631) in the SRI 400-m towing tank. Ship Performance Division Report No. 00-004-01, The Ship Research Institute of Japan, Mitaka, Tokyo
- Kume K, Ukon Y, Fujisawa J, et al (2000) Uncertainty analysis for the KCS model (SRI M.S.No.631) test in the SRI 400-m towing tank. Ship Performance Division Report No. 00-008-01, The Ship Research Institute of Japan, Mitaka, Tokyo
- Stern F, Wilson RV, Coleman HW, et al (2001) Comprehensive approach to verification and validation of CFD simulations—part 1: methodology and procedures. J Fluids Eng 123:793–802
- Wilson RV, Stern F, Coleman HW, et al (2001) Comprehensive approach to verification and validation of CFD simulations—part 2: application for RANS simulation of a cargo/container ship. J Fluids Eng 123:803–810
- Wilson RV, Shao J, Stern F (2004) Discussion: criticisms of the "correction factor" verification method [Roache P (2003) ASME J Fluids Eng 125:732–733]. ASME J Fluids Eng 126:704–706
- Wilson RV, Paterson E, Stern F (2000) Verification and validation for RANS simulation of a naval combatant. In: Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics. Chalmers University of Technology, Gothenburg, Sweden
- Tahara Y (1999) Wave influences on viscous flow around a ship in steady yaw motion. J Soc Naval Archit Jpn 186:157– 168
- Tahara Y, Ando J (2000) Comparison of CFD and EFD for KCS container ship in without/with propeller conditions. In:

Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics. Chalmers University of Technology, Gothenburg, Sweden

- Tahara Y, Ando J, Himeno Y (2001) CFD-based optimization of tanker stern form: minimization of delivered horsepower using self-propulsion simulator. In: Practical design of ships and other floating structures. Shanghai, China, pp 719– 724
- Wilson R, Carrica P, Stern F (2004) A single-phase level set method with application to breaking waves and forward speed diffraction problem. 25th Symposium on Naval Hydrodynamics, August 8–13, St. John's, Newfoundland and Labrador, Canada (CD-ROM)
- Wilson R, Carrica P, Stern F (2005) RANS simulation of a container ship using a single-phase level set method with overset grids. CFD Workshop Tokyo 2005, 9–11 March, Tokyo, Japan, pp 546–551
- Carrica P, Wilson RV, Stern F (2006) Unsteady RANS simulation of the ship forward speed diffraction problem. Comput Fluids 35:545–570
- 22. Carrica P, Wilson RV, Stern F (2006) An unsteady singlephase level set method for viscous free surface flows. Int J Numer Methods Fluids (in press)
- Rhee S, Stern F (2001) Unsteady RANS method for surface ship boundary layer and wake and wave field. Int J Numer Methods Fluids 37:445–478
- Wilson R, Stern F (2002) Unsteady RANS simulation of a surface combatant with roll motion. 24th Symposium on Naval Hydrodynamics, July 8–13, Fukuoka, Japan (CD-ROM)
- Weymouth G, Wilson RV, Stern F (2005) RANS CFD predictions of pitch and heave ship motions in head seas. J Ship Res 49:80–97
- 26. Campana EF, Peri D, Tahara Y, et al (2004) Comparison and validation of CFD-based local optimization methods for surface combatant bow. 25th Symposium on Naval Hydrodynamics, August 8–13, St. John's, Newfoundland and Labrador, Canada (CD-ROM)
- Tahara Y, Wilson RV, Carrica P (2005) Comparison of free surface capturing and tracking approaches in application to modern container ship and prognosis for extension to selfpropulsion simulator. CFD Workshop Tokyo 2005, March 9– 11, Tokyo, Japan, pp 604–611
- Tahara Y, Katsui T, Himeno Y (2004) Development of simulation-based design for ship hydrodynamics and fluid engineering. 4th Conference for New Ship and Marine Technology, October 26–29, Shanghai, pp 1–13
- Nakatake K (1981) A practical method to calculate propulsive performance of ships. Mem Faculty Eng Kyushu Univ 41:87– 122

ORIGINAL ARTICLE

Computational fluid dynamics-based multiobjective optimization of a surface combatant using a global optimization method

Yusuke Tahara · Daniele Peri Emilio Fortunato Campana · Frederick Stern

Received: May 12, 2007 / Accepted: November 14, 2007 © JASNAOE 2008

Abstract The main objective of this article is to describe the development of two advanced multiobjective optimization methods based on derivative-free techniques and complex computational fluid dynamics (CFD) analysis. Alternatives for the geometry and mesh manipulation techniques are also described. Emphasis is on advanced strategies for the use of computer resource-intensive CFD solvers in the optimization process: indeed, two up-to-date free surface-fitting Reynolds-averaged Navier-Stokes equation solvers are used as analysis tools for the evaluation of the objective function and functional constraints. The two optimization methods are realized and demonstrated on a real design problem: the optimization of the entire hull form of a surface combatant, the David Taylor Model Basin-Model 5415. Realistic functional and geometrical constraints for preventing unfeasible results and to get a final meaningful design are enforced and discussed. Finally, a recently proposed verification and validation methodology is applied to assess uncertainties and errors in simulation-based optimization, based on the differences between the numerically predicted improvement of the objective function and the actual improvement measured in a dedicated experimental campaign. The optimized model demon-

Y. Tahara (⊠) Department of Marine System Engineering, Osaka Prefecture University, 1-1 Gakuen-cho, Osaka 599-8531, Japan e-mail: tahara@marine.osakafu-u.ac.jp	
D. Peri · E.F. Campana NSEAN—Italian Ship Model Basin, Rome, Italy	
F. Stern THR—Hydroscience and Engineering, University of Iowa, Towa, USA	

strates improved characteristics beyond the numerical and experimental uncertainty, confirming the validity of the simulation-based design frameworks.

Key words Simulation based design · Shape optimization · Multi-objective optimization · Derivative-free optimization · Verification and validation

List of symbols

$B_n(t)$	Bezier curve of degree n
$D_{\rm P}, D_{\rm O}$	Experimental value of the
	parent and the optimized
	designs
E_{Δ}	Difference between the
	measured and the expected
	improvements
$f, f_0,$ etc.	Fitness function
$\vec{F} = (F_1, F_2, \ldots, F_N)^T$	Multiobjective functions
$F_n = U / \sqrt{g L_{PP}}$	Froude number
$\bar{f}(t)$	Average fitness of a population
g	Gravitational acceleration
h, g	Equality and inequality
	constraint functions
L_{PP}	Ship length
Μ	Number of design variables
т	Number of processors
n	Population size
Ν	Number of multiobjective
	functions
$N_{i,p}, N_{i,q}$	Normalized B-spline basis
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	functions of degree p and q
v, q	Number of equality and
. –	inequality constraint functions

P, Q, R	Grid clustering and stretching
	functions
$P_{i,j}$	Location vectors of
	nonuniform rational B-spline
	(NURBS) control points
$P^{\circ}, \delta P$	Original and displacement
	location vectors
ľ	Penalty parameter
$R_n = UL_{PP}/v$	Reynolds number
R_P	Pareto ranking
R_T	Total resistance
S(u, v)	3D surface defined by NURBS
S, D	Simulation value, and date
,	value
$S_{\rm P}, S_{\rm O}$	Numerical simulation value of
~,~0	the parent and the optimized
	designs
II	Ship speed
	NURBS parameters
II II etc	Uncertainty
$\vec{v}_{\rm S}, \vec{v}_{\rm SN}, \vec{c} \vec{c}$. $\vec{v} (\vec{x} \rightarrow)$	Velocity components
$u(x \rightarrow)$	normalized by ship speed U
141	Weights
$W_{i,j}$ V V Z	Nondimonsional Cartasian
Λ, I, L	Nondimensional Cartesian
	coordinates, normalized by snip
\vec{x} (,,) T	length L_{pp}
$\begin{array}{c} x = (x_1, x_2, \dots, x_M) \\ \overrightarrow{} \overrightarrow{} \end{array}$	Design variables
x, y	Points in the multiobjective
0 = k+1	function space
$x^{\circ}, x^{\circ \circ \circ}$	Original and new Bezier
	patched surface
$x_i(t)$	Frequency of genotype B_i at
	generation t
x_i^u, x_i^l	Upper and lower bounds of
	design variables
χ	Subset χ of the <i>M</i> -dimensional
	real space \mathfrak{R}^{M}
$\Delta_{\rm S}, \Delta_{\rm D}$	Differences in value between
	the parent and the optimized
	designs
λ^*	Nondimensional wavelength
ν	Kinematic viscosity
ρ	Density of water
$\xi^{1}, \xi^{2}, \xi^{3}$	Computational coordinates
ξ_{3}, ξ_{5}	Heave and pitch peaks of
	response amplitude operator
	for head seas for $\lambda^* \ge 0.4$

1 Introduction

This article describes the development of two different multiobjective optimization methods, their application to the optimization of a surface combatant ship [the David Taylor Model Basin (DTMB) Model 5415], and the experimental verification to assess the success of the optimization. This work is based on a close interaction among IIHR—Hydroscience and Engineering of the University of Iowa, the Italian Ship Model Basin (INSEAN), and the Osaka Prefecture University (OPU) in the framework of their respective Office of Naval Research (ONR) Naval International Cooperative Opportunities in Science and Technology Program (NICOP) research projects (2002–2005 and 2005–2008). The work reported by Campana et al.¹ was precursory to the present study; results for a single-objective optimization were presented and the methods were successfully validated.

Although the use of reliable and validated computational fluid dynamics (CFD) solvers is rapidly becoming a common practice in the advanced ship design process, these methods of analysis are often adopted as a shortcut to reduce the standard activities carried out in towing tank facilities, which are considered to be both too long and too expensive to be fully included in the design process. However, mature CFD analysis can be used in a more fruitful way, and the next objective is to explore its usefulness in simulation-based design (SBD). Such approaches have been developed by the present authors²⁻¹⁰ in an effort to deal with complex, real-life design problems and constraints, and include a severe evaluation of the results through model test verification.¹ Beside the studies cited above, many recent applications of CFD-based optimization witness that optimal shape design is receiving growing consideration in the naval hydrodynamics community and is finally starting to close the gap with other fields (automotive and aeronautical engineering, for example) in which SBD frameworks are a real tool in the design of complex systems.^{11–15}

Typical ship design problems involve multiple objectives.¹⁶ For instance, the goals of the design process include resistance reduction, lower hydrodynamic noise, minimal bow wave height, and the reduced amplitude and acceleration of particular motions. In addition, ship designers may also be interested enhancing certain quantities related to the engine power or to the maintenance costs. Unfortunately, the improvement of a specific aspect of the global design usually causes the worsening of some others, and the best approach is not to combine all the objectives into a single one (so-called *scalarization*) but to keep the multicriteria nature of the problem and to rely on the Pareto optimality concept.¹⁷

Furthermore, real-life ship design problems are nonlinear and also nonconvex. Indeed, the enforcement of nonlinear constraints generally leads to a nonconvex optimization problem, i.e., the feasible solution set might
be a nonconvex set, often composed of the sum of disconnected nonconvex subsets, therefore excluding the possibility of using local optimization algorithms. Instead, a correct solution approach must adopt a global optimization scheme, since a local optimization algorithm cannot jump across the gaps created by the nonlinear constraints to reach more promising feasible regions of the design space. In addition, local optimizers can also be easily trapped by suboptimal solutions that may be rather close to the starting point (i.e., the initial design).

For these reasons, our focus is on the development of global multiobjective optimization procedures for ship design, with an approach similar to the strategy presented in Campana et al.¹ for single-objective problems. Two SBD environments are finally developed, involving the combination of flow simulation, geometry modeling, and optimization scheme required for the design of complex systems. All the functional components needed to support SBD are described. The CFD analysis tools are two high-fidelity, free-surface Reynolds-averaged Navier-Stokes (RANS) equation solvers, namely CFDSHIP-Iowa^{18,19} and MGShip,²⁰ classified in the Gothenburg 2000 workshop as the best two codes for the DTMB Model 5415 test case.²¹ Indeed, the use of a RANS equation solver is essential in the present design problem, in which specific flow features related to sonar dome vortices and the wet-transom stern must accurately be predicted. The two SBDs differ also in terms of the multiobjective optimization strategy: one explores the capabilities of the parallel-computing-based multiobjective genetic algorithm (MOGA), whereas the other is based on a derivative-free uniform covering (UNICO) approach, coupled with a variable-fidelity technique. The geometry modeling and modification method is also different in the two SBD environments. In order to evaluate the relative performance of the methods and the validity of the results, optimizations were performed on the surface combatant DTMB Model 5415, allowing the entire hull form to be modified. Real-life constraints were enforced during the optimization to provide realistic optimal designs.

Finally, to assess the success of the optimization process, a dedicated experimental campaign was carried out on both the original and the optimized models. The experimental data were used in a validation procedure recently proposed by Campana et al.¹ and based on the analysis of the trend of the objective function to be minimized. This procedure represents an extension of the one proposed by Stern et al.²² and Wilson et al.²³ for single numerical simulation and builds on the validation approach²⁴ for trends by adding verification considerations. Indeed, due to the inherent uncertainties in simu-

lations and experiments, the success of an optimization process is more appropriately based on trends than absolute values. The results reported below demonstrate the success of both SBD environments.

2 CFD-based multiobjective optimization methods

The CFD-based optimization methods for shape design consist of three main components (Fig. 1), i.e., a method to solve the nonlinear programming (NLP) problem, a geometry modeling method, and a CFD solver used as an analysis tool to return the value of the objective function and of functional constraints. In the present study, two approaches are investigated for each component. These are the MOGA versus the derivative-free UNICO with variable-fidelity approaches, computer-aided design (CAD)-based versus CAD-free surface modeling approaches, and two-leading RANS codes that were discussed in the Gothenburg 2000 workshop,²¹ i.e., CFDSHIP-Iowa versus MGShip. Finally, with these elements, two different multiobjective optimization methods were realized. The components are basically extended version of those used in Campana et al.¹ for the present application of multiobjective optimization. Some details have already been described by Campana et al.¹, and so only an overview and the new features are described here.

2.1 Optimizers

Shape design optimization is typically formulated in the framework of a NLP problem. For a general expression of an *N*-objective function optimization problem in ship hydrodynamics, the mathematical formulation assembles all the design variables x_1, x_2, \ldots, x_M in a vector $\vec{x} = (x_1, x_2, \ldots, x_M)^T$ belonging to a subset χ of the *M*-dimensional real space \Re , i.e., $x \in \chi \subseteq \Re$ (upper x_i^u and lower x_i^l bounds are typically enforced onto the design



Fig. 1. Basic elements of a computational fluid dynamics (*CFD*)based optimization environment

variables). The objective of the optimization $\vec{F} = (F_1, F_2, \ldots, F_N)^T$ and the equality and inequality constraints h and g are functions of the design variables \bar{x} and of the state of the system $\vec{u}(\bar{x})$. A general form for constrained NLP problems is then used to find the particular vector \tilde{x} in the subset χ which solves the following:

$$\operatorname{Min:} \begin{cases}
[F_{1}(\vec{x}, \vec{u}(\vec{x}); R_{n}, F_{n})]_{R_{n}, F_{n}} \\
[F_{2}(\vec{x}, \vec{u}(\vec{x}); R_{n}, F_{n})]_{R_{n}, F_{n}} \\
\bullet & , x \in \chi \subseteq \Re^{M} \\
\bullet \\
[F_{N}(\vec{x}, \vec{u}(\vec{x}); R_{n}, F_{n})]_{R_{n}, F_{n}}
\end{cases} (1)$$

Subject to
$$h_j(\vec{x}) = 0$$
 $j = 1, \dots, p$
 $g_j(\vec{x}) \le 0$ $j = 1, \dots, q$
 $x_i^l \le x_i \le x_i^u$ $i = 1, \dots, M$

where F_n and R_n are the Froude number and the Reynolds number, respectively. The physical state of the system is numerically evaluated by solving a system of partial differential equations of the general form $A(\bar{x}, \bar{u}(\bar{x})) = 0$, e.g., RANS equations in the present study. In the following analysis, two alternatives to solving the above NLP problem are described.

Optimizer-A: multiobjective genetic algorithm (MOGA) approach

The adopted scheme is an extended genetic algorithm (GA) for multiobjective optimization problems, i.e., MOGA.^{25–27} The basic procedure follows that of a GA, as illustrated in Fig. 2, i.e., (i) generation of an initial population of individuals in a random manner; (ii) decoding and evaluation of some predefined quality criterion, referred to as the fitness f; (iii) selection of individuals based on a probability proportional to their relative fitness; (iv) crossover and mutation. The steps (ii) through (iv) are repeated for the designated number of generations. For crossover and mutation operations, the ratios must be given as system parameters.

The extension of GA to MOGA is straightforward. The main goal is to detect a uniformly distributed globally Pareto-optimal front set, which is defined thus: the nondominated set of the entire feasible search space is the globally Pareto-optimal set.²⁷ In order to make the conditions of Pareto optimality mathematically rigorous, we state that a vector \vec{x} is particularly less than \vec{y} , symbolically $\vec{x} < P \vec{y}$, when the following condition holds: $(\vec{x} < P \vec{y}) \Leftrightarrow (\forall i)(x_i \le y_i) \land (\exists i)(x_i < y_i)$. Under this circumstance, we say that point \vec{x} dominates point \vec{y} . If a point is not dominated by any other, we say that



Fig. 2. High-performance parallel-computing architecture for multiprocess genetic algorithm. *Gen.*, generation



Fig. 3. Pareto ranking (R_p) operation used in multiobjective genetic algorithm (MOGA) optimizer-A

it is nondominated, or noninferior. The basic definition is used to find noninferior points in MOGA in association with the Pareto-ranking technique in the present study. At each generation, higher fitness f_0 is given to individuals of higher Pareto ranking R_P (see Fig. 3 for the two-objective function case), i.e., $f_0 = 1/R_P$. The functional constraints are accounted for by using a penalty function approach, which artificially lowers the fitness if the constraints are violated and is expressed as:

$$f = f_0 - r \left[\sum_{j=1}^p \left| (h_j \vec{x}) \right| + \sum_{j=1}^q \left| \max\left\{ 0, g_j(\vec{x}) \right\} \right| \right]$$
(2)

where *r* is a penalty parameter.

The present GA and MOGA schemes can be used in both serial- and parallel-computation modes. The authors developed parallel GA and MOGA by introducing a message-passing interface (MPI) protocol,^{1,28,29} and demonstrated single- and multiobjective optimizations. The paralleled part of the algorithm is indicated in Fig. 2, i.e., processor 0 controls the overall GA procedure, and processors 1 through m, where m is the population number, simultaneously execute the CFD method, i.e., evaluate f in the figure. It was shown in earlier work²⁹ that the present parallel MOGA indicates satisfactory results, e.g., the introduction of a parallel architecture effectively enhanced computational speed and the accuracy was found to be equivalent to one of the advanced derivative-based multiobjective optimization schemes.

Optimizer-B: uniform covering (UNICO) approach with variable fidelity

This scheme belongs to the class of deterministic optimizers and to the family of covering methods with some features similar to adaptive clustering covering.³⁰ Apart from those special cases in which a specific knowledge on the objective functions is available (e.g., they are convex or monotonically decreasing), the solution of the optimization problem requires an exhaustive exploitation of the design space. This usually involves a twophase approach: on one hand, a *global* search identifies suitable subsets where promising candidates of global minima (supposed to exist) are confined. Then, an efficient local search provides accurate approximations of each candidate by exploring the corresponding subset. Thus, the resulting algorithm includes the computational burden of both the *global* and the *local* phase. UNICO is essentially based on a uniform search of the design parameter space. The algorithm consists of two main stages: (i) a global exploration of the design space, trying to locate optimal regions where attractive solutions might be found and (ii) a local refinement phase, where best configurations (according to a decision maker) are grouped in clusters and then locally optimized with a multiobjective gradient-based technique. In order to reduce the computational burden, the use of a metamodel (see, for example, Peri and Campana^{8,9} and Paterson et al.¹⁹) and of the variable-fidelity concept is adopted.

The main steps of the algorithm may be summarized as follows:

- 1. *Initial exploration of the design space*. An orthogonal array technique³¹ is adopted for the initial exploration of the design space and trial points are distributed.
- 2. *Model identification from CFD results*. Trial points are evaluated using CFD for the construction of metamodels (one for each objective function, see, for example, Peri and Campana¹⁰).
- 3. Search in the design variable space. New trial designs are distributed in the design variable space by using a uniformly distributed sequence in multidimentional parameter space known as $LP\tau$ sequence.³²
- 4. *Derive the feasible set*. Enforcing the geometrical (e.g., box constraints on the body plan) and functional constraints (e.g., ship stability), a large part of these trial points is eliminated. Only the feasible solutions are stored and the feasible set is derived.
- 5. *Identify the Pareto front*. Analyze all the feasible solutions using the metamodels and find all the solutions belonging to the Pareto front.
- 6. *Adopt a decision maker*. Select a strategy to order the designs and find the dominating solutions.
- 7. *Local refinement*. The best solutions are taken as the initial point of a multiobjective gradient step based on the metamodels (or with a scalarization of the problem, according to the decision maker choice).
- 8. *Verification*. The best solution is evaluated using high-fidelity CFD solvers. The new solution will be added to the metamodel for its improvement.
- 9. *Clustering*. Pareto solutions are then clustered around dominating solutions identified by the decision maker and a reduced number of sets is obtained.
- 10. *Refinement*. A refinement step is taken, distributing new trial designs around the center of the clusters: new trial designs are uniformly distributed with smaller LPt grids centered around the clusters.
- 11. *Iteration*. Go to step 4 until no more regions of attraction are founded.
- 12. *Final local search*. Finally, the best solution is locally optimized using the variable fidelity technique reported in Campana et al.¹

At step 9, several clustering techniques may be adopted (see, for example, Törn and Žilinskas³³). To reduce the computational costs involved in the evaluation of the gradient components during the local optimization phase, the variable-fidelity approach is important. Indeed, the use of high-fidelity models is expensive in the iterative optimization procedure. On the other hand, the use of corresponding lower-fidelity models alone does not guarantee improvement to a higher-fidelity design. The idea of using computational models of varying fidel-

Deringer 2

ity has a long history in engineering design, and more details are given in Campana et al.¹

On the other hand, the LP τ grid approach is a rational way to uniformly distribute points inside a prescribed portion of the design space. A sequence of points uniformly spaced is easily determined by using tabular data reported in Statnikov and Matusov.³² The criteria governing the choice of the position of the sampling point in space, and the tabular data as well, is that for every prescribed volume (smaller than the analyzing region) placed into the investigated portion of the design space; the number of included points do not change with the position of the control region. The limitations coming from the tabular data³² are: space dimension less than 20 and number of points less than 65 536. These limits are fully compatible with the class of problems we are going to solve.

2.2 Geometry and grid manipulation

Tools for geometry modeling (and its necessary sequel, automatic grid deformation) are another relevant SBD component. An efficient and flexible way to modify the geometry of the body is necessary for a full investigation of the design variable space and a successful optimization. Techniques should be versatile enough to describe a broad variety of complex three-dimensional (3D) configurations and be sufficiently compact so as to use as few variables as possible. Once the optimization algorithm obtains the vector with the new design variable values, we have to spread the deformation over the body surface and the computational volume grid. Flexible methods are the superposition of several basic forms (morphing techniques) or the expansion/reduction of basic geometry. Another capable method for geometry modeling is through application of CAD systems. In the present study, two approaches are investigated, i.e. a CAD-based approach and an additive perturbation (CAD-free) method.

Geometry method-A: CAD-based approach

To modify the ship geometry, a CAD-based hull form modification method was adopted. Two approaches are possible, i.e., CAD direct control and CAD emulation approaches. Both approaches were successfully demonstrated by the present authors.^{1,3,5} In the present work, the CAD emulation approach is used. As shown in Fig. 4, a module is implemented in order to emulate CAD operation handling with mathematical surface modeling. This approach offers an advantage over the CAD direct control approach for complete independence from the CAD system, i.e., designers are able to use any CAD



Fig. 4. Implementation of computer-aided design (*CAD*)-based hull form modification in the optimization environment. *IGES*, initial graphics exchange specification

system and give/receive initial/optimized hull form geometry in initial graphics exchange specification (IGES) format data. For example, the nonuniform rational Bspline (NURBS) is widely used in CAD systems for hullform design as IGES entity 128. A NURBS surface is mathematically given by:

$$\mathbf{S}(u,v) = \frac{\sum_{i=0}^{n} \sum_{j=0}^{m} N_{i,p}(u) N_{j,q}(v) w_{i,j} \mathbf{P}_{i,j}}{\sum_{i=0}^{n} \sum_{j=0}^{m} N_{i,p}(u) N_{j,q}(v) w_{i,j}}$$
(3)

where *u* and *v* are parameters; $N_{i,p}$ and $N_{i,q}$ are normalized B-spline basis functions of degree *p* and *q* in the *u* and *v* directions, respectively; $P_{i,j}$ are location vectors of the control points; and $w_{i,j}$ are weights. Finally, the surface is defined by $(n + 1) \times (m + 1)$ control points, weights, and knot vectors of n + p + 2 and m + q + 2 elements in the *u* and *v* directions, respectively. A modified surface is defined in correspondence to new location vectors Pⁿ, so that:

$$\mathbf{P}_{i,j}^n = \mathbf{P}_{i,j}^o + \delta \mathbf{P}_{i,j} \tag{4}$$

where \mathbf{P}° and $\delta \mathbf{P}$ are the original and displacement location vectors. $\delta \mathbf{P}$ can be the design variables of the optimization problem.

Geometry method-B: CAD-free approach using free form deformation (FFD)

With this approach, the use of CAD or parameterization of the hull surface is avoided and the deformation of the shape is defined and controlled by using a few control points, much fewer than the number of nodes used for the discretization adopted in the flow analysis. Mesh movement through the optimization cycles is enforced in an explicit and simple way: details can be found in Sederberg and Parry.³⁴ The original ship geometry, usually given in a CAD format (IGES file), is easily translated into a standard ASCII format (e.g., plot3d) that constitutes the input for the optimization code. The final geometry (the output of the optimization code) is easily translated back into the IGES format, ready for the milling machine. The first step of the procedure is to modify the hull surface according to the needs of the optimizer. The selected portion of the ship hull to be modified is embedded into a control region and its deformation, obtained by moving a limited number of control points, is obtained by moving the ship hull points to form the computational grid with a prescribed law. The number and position of the control points, as well as the portion of the ship subject to a shape modification, can be changed in an easy and flexible way, depending onto the details of the assigned problem. Figure 5 illustrates hull definition through application of this approach.



Fig. 5. Free form deformation (FFD) is the adopted strategy for hull parameterization in geometry method B

Once the ship surface is modified, the same volume grid regeneration technique is used as in CAD-based approaches.

The underlying idea is not to model the body's shape with Bezier surfaces (this will lead to inevitable problems when dealing with nonsmooth objects) but just to model the deformation d^k of the original shape x^0 . The following polynomial curve in 1D space:

$$B_{n}(t) = \sum_{i=0}^{n} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} p_{i} \quad \text{where} \quad \left(\frac{n}{i}\right) = \frac{n!}{n!(n-i)!}$$
(5)

is a Bezier curve of degree *n*. Here *t* is the parameter, varying from 0 to 1, adopted for the parametric expression of the curve, and p_i is the *i*th weight of the polynomial curve, *n* being also the number of control points of the curve. Two such polynomials may be adopted in order to define a curve in 2D space: in this case, two different sets of parameters px_i and py_i represent the position of the control point of the Bezier curve in 2D space, and the equation of the curve will be expressed in the parametric form:

$$\begin{cases} x(t) = \sum_{i=0}^{n} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} p x_{i} \\ y(t) = \sum_{i=0}^{n} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} p y_{i} \end{cases}$$
(6)

A Bezier surface in 3D space comes from an extension of the Bezier curve by cross product. Here a single parameter is no longer sufficient for the surface description, and a structured grid of control points in 3D space is required for a generic representation of the surface. This leads to the expression:

$$\begin{cases} x(t,s) = \sum_{i=0}^{n} \sum_{j=0}^{m} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} \left(\frac{m}{j}\right) (1-s)^{m-j} s^{j} p x_{ij} \\ y(t,s) = \sum_{i=0}^{n} \sum_{j=0}^{m} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} \left(\frac{m}{j}\right) (1-s)^{m-j} s^{j} p y_{ij} \\ z(t,s) = \sum_{i=0}^{n} \sum_{j=0}^{m} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} \left(\frac{m}{j}\right) (1-s)^{m-j} s^{j} p z_{ij} \end{cases}$$
(7)

where t and s are the parameters for the parametric expression of the surface, and px, py, and pz are the $n \times m$ matrix of the control point coordinates. Based on this framework, a deformation tool was proposed by Sederberg and Parry,³⁴ in which a further space extension of the Bezier polynomial curve was adopted. Here the expression of a scalar function v(t, s, q) inside a unit cube is given:

$$v(t, s, q) = \sum_{i=0}^{n} \sum_{j=0}^{m} \sum_{k=0}^{l} \left(\frac{n}{i}\right) (1-t)^{n-i} t^{i} \left(\frac{m}{j}\right) \times (1-s)^{m-j} s^{j} \left(\frac{l}{k}\right) (1-q)^{l-k} q^{k} p_{ijk}$$
(8)

Once a parallelepiped is located in 3D space, the parameters for the parametric expression (t, s, q) are easily obtained by nondimensionalizing the physical coordinates. Three different functions, representing the deformation to be applied along the three directions of Cartesian space, can be adopted in order to enforce a complete modification of the space originally located inside the parallelepiped, along all three dimensions: a deformation vector is obtained this way. As a consequence, if d^k is the deformation vector computed in correspondence to the grid point x^k , the new position ξ^k for that grid point is obtained by:

$$\xi^k = x^k + d^k \tag{9}$$

The number and position of the patches and number of control points per patch can be changed, depending on the details of the assigned problem. Some control points can be frozen or can be forced to move jointly with other control points. Applications of this method can be found in the report by Peri and Campana.⁸

Grid manipulation

Once the ship surface is modified, the volume grid around the hull should change accordingly with a simple adaptive algorithm. The same grid manipulation is used in association with both methods. During the optimization, the grid is updated at every optimization cycle as the hull form is modified. If a structured grid system is used, this is accomplished by the use of an algebraic scheme to increase the computational efficiency. A similar approach was used previously by the authors;^{24,8} however, a more simplified scheme was found to be effective in the present study. The method is described in the following. After an initial grid is generated, the geometrical information is computed and stored in memory as follows:

$$\begin{cases} P = S^{1}(\xi^{1}, \xi^{2}, \xi^{3}) \\ Q = S^{2}(\xi^{1}, \xi^{2}, \xi^{3}) \\ R = S^{3}(\xi^{1}, \xi^{2}, \xi^{3}) \end{cases}$$
(10)

where *P*, *Q*, and *R* are grid clustering and stretching functions defined in the (ξ^1, ξ^2, ξ^3) directions, respectively. More specifically, they are normalized metrics of (ξ^1, ξ^2, ξ^3)

 ξ^3) coordinates, such that $0 \le S^i \le 1$, and $S^i = 0$ and $S^i = 1$ for $\xi^i = 1$ and $\xi^i = \xi^i_{max}$, respectively. The grid points for the original geometry are already defined in computational coordinates, i.e.,

$$\begin{cases} x = x_0(\xi^1, \xi^2, \xi^3) \\ y = y_0(\xi^1, \xi^2, \xi^3) \\ z = z_0(\xi^1, \xi^2, \xi^3) \end{cases}$$
(11)

and the hull surface is expressed as:

$$\begin{cases} x = x_0(\xi^1, 1, \xi^3) \\ y = y_0(\xi^1, 1, \xi^3) \\ z = z_0(\xi^1, 1, \xi^3) \end{cases} \text{ and } \begin{cases} x = x_m(\xi^1, 1, \xi^3) \\ y = y_m(\xi^1, 1, \xi^3) \\ z = z_m(\xi^1, 1, \xi^3) \end{cases}$$
(12)

where ξ^2 is taken to be the direction normal to the surface, and values with subscript 0 and *m* correspond to the original and modified hull surfaces. The grid points at the outer boundary are fixed and are given by:

$$\begin{cases} x = x_0(\xi^1, \xi_{\max}^2, \xi^3) \\ y = y_0(\xi^1, \xi_{\max}^2, \xi^3) \\ z = z_0(\xi^1, \xi_{\max}^2, \xi^3) \end{cases}$$
(13)

In the optimization procedure, the hull surface is modified but other computational boundaries. In earlier work,^{2,4} all grid points were relocated using *P*, *Q*, and *R* when the surface was modified, and an iterative approach was used to complete the procedure. On the other hand, a simpler grid relocation method can be applied if the modification is assumed to occur at the local scale, i.e., the method is based on Q only and is simply written as:

$$\begin{cases} x = x_0(\xi^1, \xi^2, \xi^3) + (x_m(\xi^1, 1, \xi^3)) \\ - x_0(\xi^1, 1, \xi^3))(1 - S^2(\xi^1, \xi^2, \xi^3)) \\ y = y_0(\xi^1, \xi^2, \xi^3) + (y_m(\xi^1, 1, \xi^3)) \\ - y_0(\xi^1, 1, \xi^3))(1 - S^2(\xi^1, \xi^2, \xi^3)) \\ z = z_0(\xi^1, \xi^2, \xi^3) + (z_m(\xi^1, 1, \xi^3)) \\ - z_0(\xi^1, 1, \xi^3))(1 - S^2(\xi^1, \xi^2, \xi^3)) \end{cases}$$
(14)

Although this method is relatively simple and straightforward, it was found able to maintain the grid quality at a level nearly equal to that of the original approach.

2.3 CFD methods

For an advanced fluid dynamic redesign of an existing vehicle on the base of its drag, accurate fluid dynamics analysis tools are necessary for guiding the optimizer toward improved solutions. In the present study, both for the evaluation of the objective function (total resistance R_T) and of the functional constraint on the sonar dome vortices, the use of the free-surface RANS equation solver, whose degree of reliability has constantly matured in recent years, is necessary. Two fundamental parameters (neglecting surface tension effects) come into play in these simulations, i.e., F_n and R_n .

The Gothenburg 2000 international workshop dealing with the numerical prediction of the turbulent flow around ships focused on testing three modern hull forms, among which the DTMB Model 5415 was selected to represent navy designs. Verification and validation procedures were performed and the workshop showed that total resistance, wakes, and free-surface waves might now be well predicted by some of the best codes. CFDSHIP-Iowa¹⁸ and MGShip,²⁰ the two analysis tools adopted in this study, were identified as the best two codes for the DTMB 5415 tests case.²¹

CFD Method-A: CFDSHIP-Iowa

CFDSHIP-Iowa version 3.02 (for more details see Wilson et al.¹⁸ and Tahara et al.²⁸) solves RANS equations for unsteady, three-dimensional, incompressible flow by using the higher-order upwind difference method, a projection method for velocity–pressure coupling, and the method of lines. The grid is updated at each time step to conform to both the body and free surfaces, where exact nonlinear kinematic and approximate dynamic free-surface boundary conditions are imposed. The k- ω turbulence model is used to effect the closure. This code, as well as the code mentioned below, was successfully applied to predict resistance and wave and flow fields around the surface combatant at the Gothenburg 2000 workshop.¹⁸ More information on this code can be found in the literature cited.

CFD Method-B: MGShip

Another free-surface RANS solver is referred to as MGShip.²⁰ MGShip is a multigrid (FASFMG) multiblock structured grid code that solves the RANS equations. The version adopted here uses a surface-fitting approach to compute the wave pattern. The mathematical model is based on a pseudocompressible formulation of the RANS equations, approximated in the discrete formulation by a finite volume technique. A secondorder essential non oscillating (ENO)-type scheme was adopted for nonviscous terms, while viscous fluxes were computed by a standard centered finite volume approximation. MGShip has been extensively validated and is currently used by the Italian Navy, as well as by the marine and aeronautical industries, for hydrodynamic and aerodynamic simulations. More information on MGShip may be found in the literature cited.

Approximation methods for CFD data: metamodels

For CFD method-B, metamodels are used to approximate CFD data. An overview of the approach is given in the following. In order to reduce the effort of the optimization task, an analytical surrogate model is adopted instead of the CFD model. Because this surrogate model is based on the CFD model analysis, this is a model of a model: as a consequence, these tools are usually named metamodels. The technique is divided into two different steps: in a first phase, a uniform sampling of the design space is performed, and the results coming from the CFD model are stored. The sampling data are usually called the *training set*, because the metamodel is trained using these data. Second, the metamodel is derived using the data from the training set. It is now evident how two different issues come from the two phases: (1) the distribution of points in the design space is to be determined, as is (2) the scheme for the approximation/interpolation of the training set.

In this work, an orthogonal array³¹ was applied for the selection of the training set. An orthogonal array is a subset of the points coming from a regular subdivision of a hypercube surrounding the design space. Points to be discharged are selected by orthogonality criteria of the remaining points. With this technique, a large number of points are eliminated, but the completeness of the exploration of the design space is preserved. Regarding the adopted metamodel, a really wide choice is available, starting from a simple quadratic model up to a multilayer neural network. In this application, a radial basis function network (RBFN) was selected. An RBFN has the form:

$$f(x) = \sum_{i=1}^{N} \sigma_i \varphi(\rho_i)$$
(15)

Where φ is a function of the distance between the computation point and a prescribed point in the design space (the center of the radial basis function φ) and σ_i represents the weight of the *i*th function. As a consequence, the unknowns in the problem of the determination of the optimal RBFN are the positions of the centers and the weights. If a least-squares approach is adopted for the determination of the weights, the total number of unknowns is N × NDV, if N is the number of poles of the RBFN and NDV is the dimension of the design space. A conjugate gradient method was applied for the determination of the optimal positions of the

centers. Details may be found in Lampariello and Sciandrone.³⁵

2.4 Integration of optimization components and SBD environment

Finally, the above-mentioned optimization components are integrated to yield two optimization approaches, i. e., SBD-A and SBD-B. Optimizations using these environments have been demonstrated at OPU and INSEAN, respectively. In the following sections, these approaches are summarized in association with additional information regarding the SBD environment.

SBD version A

SBD-A uses MOGA, the CAD-based and grid manipulation method, and CFD-SHIP Iowa RANS solver version 3.02. For the present application, the CAD emulation type was selected for the geometry method. The integrated system also involves the ship motion program (SMP),³⁶ based on a strip theory approach (for an outline of the approach, see for instance, Newman³⁷), to evaluate the response amplitude operators (RAOs), which are part of the objective functions and constraints described later. The computations were performed on a 64-CPU PC cluster (Intel Xeon 2.4 GHz × 64) which was recently designed by the authors and installed at OPU.²⁸

SBD version B

In the present version of SBD-B, the optimizer adopts the variable-fidelity approach. SBD-B comprises UNICO with a variable-fidelity approach, the FFD with grid manipulation described above, and the MGShip RANS solver. As was done in SBD-A, the SMP code was used to evaluate the heave and pitch RAOs. The computations were performed on an Intel Xeon processor (2.8 GHz).

3 Surface combatant entire hull optimization problem

For the complete definition of the design problem to be solved, the following fundamental items must be precisely addressed: (1) selection of an initial design to be optimized and of the extension of the modifiable region, (2) selection of the objective function to be minimized plus the number and position of the design variables, and (3) the type and quantity of the constraints of the problem. All these items are described in the following sections.

3.1 Initial design

The initial design is Model 5415, which was conceived as a preliminary design for a Navy surface combatant in around 1980. The hull geometry, available in IGES format, includes both a sonar dome and transom stern. Propulsion is provided through twin open-water propellers driven by shafts supported by struts. There is a large experimental fluid dynamics (EFD) database for Model 5415 due to an international collaborative study on EFD/CFD and uncertainty assessment between IIHR, INSEAN, and the Naval Surface Warfare Center, Carderock Division (NSWC).³⁸ The validation data includes the boundary layer and wake, longitudinal wave cuts, bow and transom wave fields, and wave breaking. In the present study, the bare hull is considered.

3.2 Objective function and functional and geometrical constraints

A complete definition of the problem, the objective function and constraints, is given in Table 1. These are basically the same as those used in the previous single-objective optimization,¹ except for extra objective functions newly defined in the present study. Three objective functions are considered, i.e., (i) the total resistance R_T at a speed of $F_n = 0.28$ and $R_n = 1.67 \times 10^7$ at model scale, (ii) the seakeeping merit function (SMF) for $F_n = 0.28$, and (iii) SMF for $F_n = 0.41$, all of which are minimized. The SMF is defined as $F = 0.5\xi_3^o/\xi_3^p + 0.5\xi_5^o/\xi_5^p$, where subscripts 3 and 5 correspond to peaks of the heave and pitch RAOs for head seas for $\lambda^* \ge 0.4$, respectively, and superscripts o and p to optimized and parent hulls, respectively. To introduce elements of a realistic design problem, ξ_3^o/ξ_3^p , ξ_5^o/ξ_5^p , and a function of the axial vorticity in a region behind the dome are chosen as functional constraints. Geometrical constraints are imposed on the design variables, the sonar device volume, the bow entry angle, and the displacement and principal dimensions of the ship. More details are given in Table 1. It must be noted that we basically use the same constraints as those used in our earlier work for a single-objective optimization,¹ so that relative comparison of the results with the present multiobjective optimization is possible.

3.3 Definition of design variables

The design variables are used to explore the design space, and changes in their values correspond to different ship designs. As a consequence, these variables are closely connected with the specific technique adopted to modify the geometry of the ship and the computational mesh.

Table 1.	Definition	of objective	functions and	constraints
----------	------------	--------------	---------------	-------------

	Туре	Definition	Notes
	Three objective functions: resistance (1) and seakeeping (2)	$F_{1} = R_{T}(Fn, Rn)$ $F_{2} = 0.5 \frac{\xi_{3}^{o}}{\xi_{3}^{p}} + 0.5 \frac{\xi_{5}^{o}}{\xi_{5}^{p}} \text{ at } F_{n} = 0.28$ $F_{3} = 0.5 \frac{\xi_{3}^{o}}{\xi_{3}^{p}} + 0.5 \frac{\xi_{5}^{o}}{\xi_{5}^{p}} \text{ at } F_{n} = 0.41$ $x \in R_{dv}^{N}$	Bare hull, fixed model Resistance at $F_n = 0.28$ and $R_n = 1.67 \times 10^7$ Seakeeping at $F_n = 0.28$ and 0.41
Functional constraints	On seakeeping	$H_{C} = \frac{\xi_{3}^{o}}{\xi_{5}^{p}} \le 1.02$ $P_{C} = \frac{\xi_{5}}{\xi_{5}^{p}} \le 1.02$	All quantities computed for $\lambda^* \ge 0.4$
	On sonar dome vortices	$\frac{\sqrt{\frac{1}{N}\sum_{i}^{N} \left(\omega_{x}^{o}\right)_{i}^{2}}}{\sqrt{\frac{1}{N}\sum_{i}^{N} \left(\omega_{x}^{o}\right)_{i}^{2}}} \leq 1, \forall i \in R_{C}$	R_c is a circular region placed at $x = -0.30$, centered at $y = 0.02$, $z = -0.07$, with radius $r = 0.018$.
Geometrical constraints	Bow entry angle Sonar dome dimension Sonar dome position Main dimensions Displacement	Maximum amplitude variation of 5° A sonar array of radius Rs and height Hs should fit inside the dome Maximum forward position L_{PP} and depth fixed Maximum variation $\pm 2\%$	2.5° per side Hs = 3 m, Rs = 5 m in ship length

 ξ_3 , peak heave response amplitude operator (RAO); ξ_5 , peak pitch RAO; o, optimized; p, parent; R_c , circular control region

For SBD-A, displacements of the NURBS control points in geometry method-A (CAD-based approach) can be the design variables of the optimization problem. In this approach, the design actions of widely used CAD tools are emulated, so that the control points that define the bow, sonar dome, and mid-after part of hull are moved in confined directions. That is, control points for the bow part move in the transverse direction only, those for the sonar dome move in transverse and longitudinal directions, and the mid-after part of hull moves in the transverse direction only. These regions are shown in Fig. 4 as regions A, B, and C, respectively. Control points are adequately grouped in order to avoid unrealistic shape modification. All points in a group move with equal displacement in the same direction. Therefore, the above-mentioned design actions are related to four design variables, which control the present hull modification.

For SBD-B, the control points of the patches used in the FFD method are the design variables of the optimization problem. For the hull shape parameterization, six design variables were used: one for the side modification of the dome, one for the longitudinal modification of the dome, and four for the side modification of the whole hull.

In previous work,¹ the definition and number of design variables needed to introduce an effective surfacemodeling scheme based on computational geometry were carefully investigated. It was found that even if a relatively small number of design variables are used, with effective grouping of control points, it is still possible to yield sufficiently large and realistic changes in the hull form. On the other hand, having too many design variables often yields unrealistic hull forms. Considering this, we decided to pursue a somewhat conservative but more realistic way of modifying the hull form by using a limited number of design variables.

4 Verification and validation (V&V) approach for SBD

In the use of CFD methods, an uncertainty assessment should be provided for the solutions and computational grid. CFD uncertainty assessment consists of verification, validation, and documentation.^{22,23} For CFD-based optimization, an additional procedure is necessary to validate the optimizer as a whole, involving performance's *trends* rather than their absolute values.¹ The approaches used in the cited references are followed here, and an overview of the approach is given in the following sections.

4.1 V&V approach for single numerical simulations

Simulation uncertainty $U_{\rm S}$ is divided into two components, one from the numerics $U_{\rm SN}$ and the other from the

modeling $U_{\rm SM}$. $U_{\rm SN}$ is estimated for both point and integral quantities and is based upon grid and iteration studies, which determine grid $U_{\rm SG}$ and iterative $U_{\rm SI}$ uncertainties. The former is evaluated by multiple grid studies and the latter can be taken from variation of the last period of oscillation. A root sum square (RSS) approach is used to combine the components and to calculate U_{SN} , i.e., $U_{SN}^2 = U_{SG}^2 + U_{SI}^2$. For CFD validation, uncertainties from both the simulation (U_{SN}) and EFD benchmark data $(U_{\rm D})$ are considered. The first step is to calculate the comparison error E, which is defined as the difference between the data D (benchmark) and the simulation prediction S, i.e., E = D - S. The validation uncertainty $U_{\rm v}$ is defined as the combination of $U_{\rm D}$ and the portion of the uncertainties in the CFD simulation that are due to numerics $U_{\rm SN}$ and which can be estimated through verification analysis, i.e., $U_V^2 = U_D^2 + U_{SN}^2$. U_V sets the level at which the validation can be achieved. The criterion for validation is

$$|E| < U_{\rm V} \tag{16}$$

that is, the combination of all the errors in D and S is smaller than U_v and then validation is said to have been achieved at the U_v level. The above-described V&V approach for a single numerical simulation was accepted as a standard procedure and practiced in recent CFD workshops (for example, see Larsson et al.²¹).

4.2 V&V approach for CFD-based optimization

We define $S_{\rm P}$ and $S_{\rm O}$ as the numerical simulation value of the parent and the optimized designs, respectively, and define the corresponding simulation uncertainty as $U_{\rm SP}$ and $U_{\rm SO}$ and the simulation numerical uncertainty as $U_{\rm SNP}$ and $U_{\rm SNO}$. We also introduce $D_{\rm P}$ as the experimental data for the parent hull and D_0 as the data for the optimized design, both with an associated experimental uncertainty $U_{\rm DP}$ and $U_{\rm DO}$. We also define $\Delta_{\rm S} = S_{\rm P} - S_{\rm O}$ and $\Delta_{\rm D} = D_{\rm P} - D_{\rm O}$ as the expected $\Delta_{\rm S}$ and the measured $\Delta_{\rm D}$ differences between the two hulls. Obviously, if the problem is one of minimization of some objective function, $\Delta_{\rm S} > 0$ and $\Delta_{\rm D} > 0$ imply improvements of the final shape with respect to the parent hull. To estimate the simulation error of the optimization process, we can build on the analysis presented in the previous section, decomposing the simulation error into a numerical and a modeling component. Assuming that the modeling errors for the parent hull and the optimized design are equal, finally, the following procedure for the V&V approach for CFD-based optimization is derived.

Verification of the trend. With reference to the uncertainties, we now need a condition that states that if the expected improvement Δ_s is greater than the simulation *numerical* noise $U_{\Delta s}$, then the optimized design is numerically verified. This can be written:

$$|\Delta_{\rm S}| > U_{\Delta \rm S} = \left(U_{\rm SP}^2 + U_{\rm SO}^2\right)^{1/2} = \left(U_{\rm SNP}^2 + U_{\rm SNO}^2\right)^{1/2}$$
(17)

Validation of the trend. In a similar manner, if the improvement measured in the experiment, Δ_D , is greater than the experimental noise $U_{\Delta D}$, then the optimized design is experimentally verified:

$$|\Delta_{\rm D}| > \left(U_{\rm DP}^2 + U_{\rm DO}^2\right)^{1/2} = U_{\Delta \rm D}$$
(18)

Optimizer's validation. The definition of the comparison error *E* has now to be modified into an *optimizer* error, E_{Δ} , focused on the trend. Hence, E_{Δ} can be defined as the difference between the measured and the expected improvements, $\Delta_{\rm D}$ and $\Delta_{\rm s}$: $E_{\Delta} = \Delta_{\rm D} - \Delta_{\rm s}$. The corresponding uncertainty equation is:

$$U_{\rm E\Delta} = \left(U_{\Delta \rm D}^2 + U_{\rm SNP}^2 + U_{\rm SNO}^2\right)^{1/2}$$
(19)

The last step is to state that if the difference between the measured and the expected improvements is less than the uncertainty $U_{\rm EA}$, that is:

$$E_{\Delta}| < U_{\rm E\Delta} \tag{20}$$

we may finally say that the optimized solution is validated at the interval $U_{\rm E\Delta}$.

4.3 V&V for the present SBD for total resistance minimization

The above-described procedure was performed for the present SBD for total resistance (R_T) minimization,¹ in which the DTMB 5415 bow optimization was considered. The information is useful in evaluating the present multiobjective optimization, since the same CFD methods and computational grid topology and density are used to evaluate R_T , defined as F_1 , one of the multiobjective functions in the present study. In particular, results related to the above-mentioned simulation numerical uncertainty are considered valid in the present application. Hence, a summary of the results is given in the following.

The experimental campaign used an existing model of DTMB 5415: the INSEAN 2340 model already adopted in former experiments.³⁸ Tank tests were carried out in towing tank no. 2 at INSEAN, and the new models were tested in the same basin. The success of the optimization

Table 2. Verification and validation (V&V) procedure for single numerical simulations relative to an integral quantity (the total resistance R_{T})

RANS solverModel $S(N)$ $D(N)$ $E(\%)$ $U_D(\%)$ $U_{sG}(\%)$ $U_{sN}(\%)$ $U_{sN}(\%)$ CFDSHIP-Iowa541545.4045.100.650.292.002.002.05415-A42.9843.451.080.532.002.002.0MGShip541546.2845.102.610.292.612.612.65415-B44.8943.483.250.301.741.741.741.74									
CFDSHIP-Iowa541545.4045.100.650.292.002.002.05415-A42.9843.451.080.532.002.002.0MGShip541546.2845.102.610.292.612.612.615415-B44.8943.483.250.301.741.741.74	RANS solver	Model	S(N)	D(N)	E(%)	$U_{\rm D}(\%)$	$U_{ m SG}(\%)$	$U_{ m SN}(\%)$	$U_{ m V}(\%)$
MGShip 5415-A 42.98 43.45 1.08 0.53 2.00 2.00 2.0 5415 46.28 45.10 2.61 0.29 2.61 2.61 2.0 5415-B 44.89 43.48 3.25 0.30 1.74 1.74 1.74	CFDSHIP-Iowa	5415	45.40	45.10	0.65	0.29	2.00	2.00	2.02
MGShip 5415 46.28 45.10 2.61 0.29 2.61 2.61 2.6 5415-B 44.89 43.48 3.25 0.30 1.74 1.74 1.74		5415-A	42.98	43.45	1.08	0.53	2.00	2.00	2.07
5415-B 44.89 43.48 3.25 0.30 1.74 1.74 1.7	MGShip	5415	46.28	45.10	2.61	0.29	2.61	2.61	2.63
	-	5415-B	44.89	43.48	3.25	0.30	1.74	1.74	1.76

RANS, Reynolds-averaged Navier–Stokes; S and D, simulated data and data benchmark values; E, comparison error; U, uncertainty; SG, simulation grid; SN, simulation numerics; V, validation

Table 3. The modified V&V procedure for CFD-based optimization applied to the optimization results obtained with simulation-based design A (SBD-A) and SBD-B

Optimization process	$\Delta_{\rm S} = S_P - S_O$	$\Delta_D = D_P - D_O$	$U_{ m \Delta S}$	$U_{\scriptscriptstyle \Delta \mathrm{D}}$	$ E_{\Delta} $	$U_{ m E\Delta}$
SBD-A	5.32	3.66	2.83	0.61	1.66	2.89
SBD-B	3.01	3.61	3.14	0.41	0.60	3.17

P, parent hull; O, optimized hull; Δ_s , Δ_D , differences in value between the parent and the optimized designs

processes was nicely confirmed by the experimental measurements. At the design speed of $F_n = 0.28$, the measured reduction in the total resistance at model scale was about 3.60% for both optimized models. The V&V analysis, summarized in the following sections and reported in Tables 2 and 3, is based on the procedure illustrated in the previous section. Initially, the numerical results are verified and validated as if they were single, isolated simulations (Table 2), then the V&V of the optimization is described (Table 3).

For SBD-A, grid convergence was studied by performing steady simulations using three computational grids with refinement $\sqrt{2}$ in each coordinate direction, i.e., fine, medium, and coarse grids. The grid sizes of those grids were 1779648, 654192, and 238760, respectively. The predicted value of the DTMB 5415 total resistance reported in Table 2 is S = 45.40 N, very close to the experimental data, giving |E| = D - S = 0.65%, while for the DTMB 5415-A the estimated total resistance is lower than the experimental value, giving |E| = D - S= 1.08%. Uncertainties in the data, U_D , are 0.29% for the original and almost double that for the 5415-A (0.53%). In both cases the comparison error \mathcal{F} is smaller than the validation uncertainty U_V , and hence the solution for total resistance is validated at the U_V level.

Given that SBD-B uses MGShip, a multigrid solver, a suite of refined grids is naturally available. Four grid sublevels were used with a refinement ratio 2 in each coordinate direction, and the three finest were used in the V&V procedure. The predicted value of the total resistance of the parent hull is S = 46.28 N (Table 2), with an error of about |E| = D - S = 2.61%. For the 5415-B, the estimated total resistance is again greater than the experimental value, and the error is larger, |E| = D - S = 3.25%. Uncertainties in the 5415-B data U_D are almost equal to those of the 5415. In one case (for the original hull) the comparison error E is smaller than the validation uncertainty U_V , and hence the solution is validated at the U_V level. The solution is not validated at the U_V level for the 5415-B model, likely due to a combination of factors: a larger error E and a smaller grid error, which reduces U_V .

The V&V procedure for CFD-based optimization, illustrated above, is applied to the present optimization results to establish the success of the optimization. The expected improvement $\Delta_{\rm s}$ estimated by SBD-A is 5.32%, whereas the actual improvement $\Delta_{\rm D}$ is smaller. However, the error E_{Δ} is smaller than the validation uncertainty $U_{\rm EA}$ and we finally may say that the optimized solution is validated for the interval $U_{\rm EA} = 2.89\%$. SBD-B predicts a smaller improvement $\Delta_{\rm s}$ of 3.01%; this is, however, very close to the actual improvement $\Delta_{\rm D}$. The error E_{Δ} is smaller than the validation uncertainty $U_{\rm EA}$, and thus the SBD-B optimized solution is validated for the interval $U_{\rm EA} = 3.17\%$.

5 Multiobjective optimized design and validation

The present optimizations follow a similar procedure to that used in previous work,¹ i.e., optimizations are performed using a coarse grid, and the results are verified using a fine grid. As shown in previous work¹ and in work by others,⁴⁻¹¹ this approach effectively reduces the

Deringer

computational effort of CFD-based optimization using the resource-intensive RANS equation solver, as long as the correct trends are predicted for both grids. The number of grids used for the coarse and fine grids are shown in Table 4. Both optimization and verification in SBD-A were carried out with the model in fixed condition, where sinkage and trim values measured for the original DTMB 5415 at $F_n = 0.28$ were given. On the other hand, SBD-B used free sinkage and trim conditions for optimization and verification.

Simulation-based design A

Optimization was performed at OPU using the scheme SBD-A. The system parameters for the GA were basically the same as those used by Campana et al.,¹ i.e., a crossover rate of 0.75, a mutation rate of 0.30, and a population size of 59. The crossover and mutation rates were determined through preliminary numerical tests. Due to limitations in the capability of available computational facilities, the Pareto-front set obtained in 20 generations is used as the final solution; however, we found that a large number of individuals were generated during the process (i.e., more than 1000 individuals) and these were more than adequate to pursue further analysis. For each generation, the RANS code was executed

 Table 4. Grids used for uncertainty analysis, optimization, and verification

	No. of grids	Refinement ratio	Coarse	Fine
CFD-A	3	$\sqrt{2}$	238 760	1 779 648
CFD-B	3	$\sqrt{2}$	253 952	1 712 128

The coarse grid was used for optimization and the fine grid was used for verification

CFD, computational fluid dynamics

once per processor using the flow field of the original hull as the initial guess, and 5000 global sweep iterations appeared to yield sufficient convergence. The wall-clock time to proceed to 20 generations was about 240 hours, i.e., 10 days, using the above-mentioned PC-cluster par-allel-computing environment. Figure 6 shows the distribution of all individuals and the detected Pareto-optimal set, in which 1180 and 113 individuals are included, respectively. Each individual on the Pareto-optimal set will be a candidate for the designer's choice.

Figure 7 shows F_1 -, F_2 -, and F_3 -minimum hulls on the Pareto-optimal set. The bow and sonar dome of the three hulls indicate similar modification trends as those found in previous work, i.e., bow and sonar dome optimization for minimum total resistance for $F_n = 0.28$. Differences are seen in the mid-after part, i.e., the maximum breadth of the hull is reduced for the F_1 minimum hull, and increased for F_2 - and F_3 -minimum hulls. All seakeeping constraints were satisfied. Finally, a hull from the Pareto-front set with a level of R_T reduction equal to that of the SBD-B optimal hull was selected (i.e., 5415-A, see Fig. 8).

Simulation-based design B

The optimization process SBD-B was carried out at INSEAN. The variable-fidelity approach described above was used in order to reduce the CPU time needed.^{7,8} The training of the metamodels for the objective functions required 25 sample points, which means 25 calls to the CFD solver. After that, 152 further solutions were allowed, giving a total number of more than 177 function evaluations (with CFD). The optimal hull (see Fig. 9) is referred to as 5415-B in the following discussions. As shown in Fig. 10, all seakeeping constraints were satisfied.

Fig. 6. Distribution of all individuals (*left*) and the detected Pareto-optimal set (*right*) obtained by simulation-based design A (SBD-A). F_1 is the total resistance objective function and F_2 and F_3 are seakeeping objective functions



Fig. 7. Comparison of the body plans for the original and optimized hull forms for SBD-A results. F_1 -minimum, F_2 minimum, and F_3 -minimum solutions are located left, center, and right, respectively. Compared to values of the original hull, sonar dome maximum widths are reduced about 20%, 18%, and 20%; dome lengths are increased about 11%, 9.5%, and 10%; and maximum breadths of the mid-after part are increased/ decreased about -1.7%, +4.5%, and +4.5%, respectively. Reductions in (F_1, F_2, F_3) are about (6%, 0.7%, 0.5%), (1%, 1.2%, 1.5%), and (1%, 1.0%, 1.8%), respectively. Bow entry angles for all hulls are reduced







5.1 Optimized hulls

Figure 11 shows the bow and sonar dome for the original and two optimized hulls. As expected, the two optimal hull forms, i.e., 5415-A and 5415-B, are not identical (see also Figs. 7–9), but show many similarities in terms of their general trends. For both results, the important modification trends in the bow region and sonar dome are the same, and similar to those in bow optimization for minimum total resistance for $F_n = 0.28$.¹ These trends are (a) the reduction of the maximum breadth of the sonar dome, (b) the extension of the sonar dome in the forward direction, and (c) reduction of the bow entry angle. In addition to these changes, the mid-after part of the hull also shows similar modification trends in both models, i.e., the maximum breadth of the hull was reduced, but the reduction in 5415-B was larger.

Fig. 9. Location of sonar device (top and left) and comparison of bow shape and body plan (top and right) between the original and optimized hull form for SBD-B. The sonar space constraint is successfully satisfied. Compared to values of the original hull, the sonar dome maximum width is reduced about 10%, the dome length is increased about 14%, and the maximum breadth of the midafter part is decreased about 4.5%. The bow entry angle is also reduced







Fig. 11. Comparison of bow and sonar dome. Original 5415 (left), 5415-A (center), and 5415-B (right)

Deringer

 Table 5. Properties of the selected optimal hulls for the present multiobjective optimizations

	Wetted surface	Displacement	L _{CB}
5415-A	-0.45	-0.92	+0.14
5415-B	-0.72	-2.00	+0.00

Values are the percentage change from the original hull 5415-A and 5415-B were obtained by using SBD-A and SBD-B, respectively

 $L_{\mbox{\tiny CB}}$ longitudinal center of buroyancy, nondimensionalised with the ship length, centered at midship and positive toward the sterm

Table 6. Predicted percentage improvement of objective functions of selected optimal hulls

	F_1	F_2	F_3
5415-A 5415-B	-5.02	-0.90	-0.90
J41J-D	-3.78	-2.55	-1.95

Reductions in the maximum breadth of the dome were about 20% and 10% for 5415-A and 5415-B, respectively. The volume reduction for both models is uniform along the dome axis and the overall shape of the new dome is similar to the original but for the width and the length. The extensions of the sonar dome are seen to be about 11% and 14% for 5415-A and 5415-B, respectively. The reduction of the bow entry angle is also obvious in both optimal hulls, and the angles are similar for both. Both models satisfy the sonar space constraints, so that the sonar fits inside the bulb (e.g., see Fig. 9 for 5415-B). Table 5 gives the properties of the optimal hulls. The reductions in the maximum breadth of the hull are 0.7% and 4.5% for 5415-A and 5415-B, respectively. The displacement and wetted surface area are also reduced for both models, i.e., those for 5415-A are reduced by 0.92% and 0.45%, respectively, and those for 5415-B by 2% and 0.72%, respectively. Due to the change in the volume distribution, the location of lcb is moved slightly backward for 5415-A, while it is kept the same for 5415-B.

5.2 Numerical results and verification

A summary of the results is shown in Table 6. The prediction is that both 5415-A and 5415-B have decreased values of R_T , i.e., by 5.02% and 3.78%, respectively. The seakeeping merit functions (SMFs) also decrease, i.e., for $F_n = 0.28$ they are reduced by 0.9% and 2.35% for 5415-A and 5415-B, respectively, and for $F_n = 0.41$ the reductions are 0.9% and 1.93%, respectively. As mentioned earlier, 5415-A was selected to have an equal level of R_T reduction as that for 5415-B at $F_n = 0.28$. This is based on the experimental work from a previous report,¹ i.e., the predicted drag reductions of the two hulls are

Table 7. Improvements in total resistance for the two optimization hull forms. The bow and sonar dome were optimized for minimum total resistance¹

	Gain—predictions	Gain-experiments
SBD-A	-5.32 (%S _{original})	-3.8 (%D _{original})
SBD-B	-3.90 (%S _{original})	-3.8 (%D _{original})

estimated to be about 4% (see correlation shown in Table 7).

It was found for both results that the reduction in SMF was mainly attributable to reduction in the maximum heave, while the pitch performance was nearly the same as that of the original hull (see Fig. 10 for 5415-B). Especially for the total resistance, the differences in predicted values are larger than the respective U_{SN} values, which yields the important fact that the improvements obtained in the present computations are meaningful.

As was the case for the bow and sonar dome optimization for minimum total resistance, reduced total resistance correlates with overall features of the computed free surface, which are shown in Figs. 12 and 13. The wave patterns for the optimized hulls are globally smoother. As is shown in the bow wave profiles in Fig. 14, for both optimized models the amplitude of the first crest is reduced. Figure 15 shows the surface pressure distribution; for both optimal hulls, improvements are seen in the pressure distribution with reduced low-values regions. The pressure varies more gently along the hull and pressure gradients are reduced in the modified region.

A functional constraint was imposed on the vorticity shed by the dome. The same definition as that for the previous R_T minimization was used for the present optimization, i.e., the axial vorticity should not exceed the original value. Figure 16 shows the vorticity distribution in the control area for the optimal and original hulls. It is clearly seen in the results that the constraint has been satisfied, i.e., the two optimal models reduce the core of the main vortex, which appears to be confined near the body surface. The reduction of dome vortices and the above-mentioned height of the bow wave crest are noteworthy positive side effects resulting from the present optimization; the former and latter are more remarkable for 5415-B and 5415-A, respectively.

5.3 Experimental validation and V&V approach for SBD

The two optimal hulls indicate close similarity in modification trends and hull 5415-B was tested using experimental measurements (see Fig. 17 for the test model).

0.1 X Optimal

Original

02

Optimal

Original

00

0.1

0.05

> 0

-0.05

-0.1

0.1

0.05

> 0

0.002

.0.004



0.4

0.2

-0.2

-04

0.4

0.2

> 0

> 0

Optimal

Original

Optimal

0.25

0.5

0.75

Fig. 13. SBD-B results: comparison of wave contours for the original (5415) and optimal (5415-B) hull forms (computed using MGShip)



Deringer

During the experimental campaign, uncertainty analysis was carried out at $F_n = 0.28$ and the uncertainty for the total resistance was found to be about 0.3%. This is partly related to the precision error of the load cell, since it decreases near the maximum measurable value, and it is also in line with previous experience. At the design speed of $F_n = 0.28$, the measured reduction in the total resistance at the model scale was about 4.75%; the error bars for the experimental campaigns were much lower than this improvement. It was also found that the seakeeping merit functions were reduced by 14.5% and 8.7% for $F_n = 0.28$ and 0.41, respectively.

Finally, the results of the V&V analysis are summarized. Tables 8 and 9 show that the results are similar in form to those for the single-objective optimization, i.e., Tables 2 and 3; however, for the present case, evaluations were made for three objective functions. For F_1 , uncertainties in the data U_D are 0.29% for the original and almost double that for 5415-B (0.53%). In both cases the comparison error E is smaller than the validation uncertainty U_V , hence the solution for the total resistance is validated at the U_V level. The expected improvement Δ_S estimated by SBD-B is 3.78% and is close to the actual improvement Δ_D (4.75%). The error E_{Δ} (0.97%) is



Fig. 15. Comparison of surface pressure (Cp) contours near the bow for the original and optimal hull forms: upper, original 5415 (computed with CFDSHIP Iowa) versus 5415-A, lower, original 5415 (computed with MGShip)



Cp contours (Original)

Fig. 16. Comparison of axial vorticity contours between the original and optimal hull forms: original 5415 versus 5415-A computed with CFDSHIP-Iowa (left), original 5415 versus 5415-B computed with MGShip (right)

smaller than the validation uncertainty $U_{\rm EA}$ (3.16%) and so the SBD-B optimized solution is validated for the interval $U_{\rm FA}$.

Some of the cells in Tables 8 and 9 are empty. This is because the uncertainties in the data $\Delta_{\rm D},~U_{\rm SG},~U_{\rm SN},$ and $U_{\rm V}$ are not available for F_2 and F_3 due to the nature of the seakeeping test itself. In fact, all the seakeeping quantities were obtained from the statistical analysis of a very long time relief, obtained as the sum of a large number of successive tests. As a consequence, the production of a set of separate time series, to be analyzed in the same way as for the resistance tests, is too expensive and it was not produced during this project. As a consequence, there is no possibility of deriving the uncertainty of the experimental data from a single sample. On the other hand, the order of magnitude of the observed gain (10%) is reasonably far away from the precision limit of these kinds of tests.

113

Cp contours (Optimal)

Table 8. V&V procedure forsingle numerical simulationsrelative to an integralquantity

Objective function	Model	S(N)	D(N)	E(%)	$U_{\mathrm{D}}(\%)$	$U_{\rm SG}(\%)$	$U_{\rm SN}(\%)$	$U_{\rm V}(\%)$
$\overline{F_1}$	5415 5415-B	46.28 44.53	45.10 42.95	2.61 3.67	0.29 0.30	2.61 1.74	2.61 1.74	2.63 1.76
F_2	5415 D	1.0	1.0		_		_	_
F_3	5415-В 5415	0.976 1.0	0.855	14.1	_		_	_
	5415-B	0.980	0.913	7.3	—			



Fig. 17. Experimental model of hull 5415-B

6 Conclusions

Two different basic SBD versions were developed and tested in a nonlinear, constrained, multiobjective optimization problem, in which three objective functions to be minimized were simultaneously considered, i.e., the total resistance for $F_n = 0.28$ and seakeeping merit functions for $F_n = 0.28$ and 0.41. The two SBD versions were applied to the optimization of the entire hull form of a surface combatant, DTMB Model 5415. Realistic functional and geometrical constraints for preventing unfeasible results and to get a final meaningful design were enforced and discussed.

Both SBD versions were found to yield satisfactory results. Experimental tests were carried out on the final optimized model, and the validity of the optimization processes was successfully confirmed. Improvements were obtained while other important qualities, e.g., sonar dome vortices, were preserved. Several noteworthy positive side effects are indicated for the optimal hulls, e.g., a significant decrease in sonar dome vortices and the height of the bow wave crest. These results, along with conclusions in precursory work for bow optimization for minimum total resistance, prove the validity of the methods developed in the present IIHR/INSEAN/OPU joint NICOP research project, in which free-surfacefitting RANS solvers were integrated with a multiobjec-

 Table 9. The modified V&V procedure for CFD-based optimization applied at the optimization results obtained with SBD-B

Objective function	$\Delta_{\rm S} = S_P - S_O$	$\Delta_D = D_P - D_O$	$U_{\Delta \mathrm{S}}$	$U_{\Delta \mathrm{D}}$	$ E_{\Delta} $	$U_{ m E\Delta}$
$\overline{F_1}$	3.78	4.75	3.14	0.41	0.97	3.16
$\vec{F_2}$	2.35	14.5			12.2	
$\tilde{F_3}$	1.93	8.7			6.77	

tive optimization cycle and an advanced hull surface modeling and modification scheme.

Nevertheless, several limitations of the present work must be stated. The main achievement of the present work is still limited to system development and demonstration of the capability, which explains the use of a relatively simplified problem definition and design conditions, and a limited number of generations used in the optimization procedure. The scheme introduced for hull form modification can be improved to allow a more flexible surface modeling approach that includes more design variables, which will be able to generate even lessconservative designs. The decision space can also be expanded, so that the capability of the present global optimization schemes will be more strongly emphasized. In spite of these shortcomings, we still consider that our approach can produce promising results and the limitations may eventually be eliminated as the simplification of the problem definition and design conditions are eliminated, more powerful computation facilities becomes available, and more advanced geometry modeling schemes are introduced.

In addition, extension of the problem and future research directions will involve application to high-speed multihull ships in association with the development and adoption of more advanced global optimization algorithms and CFD methods, which include advanced free-surface modeling and overset grid capability.³⁹⁻⁴¹ Such an attempt is already in progress.⁴²

The development of a SBD framework that combines highly costly analysis tools and global optimization algorithms may appear to be a paradox, but design engineers of marine, aeronautical, and automotive transport systems are very much inclined to take this direction. Indeed, the margin for design improvements is continuously narrowing because design engineers are producing near-optimal configurations in many industrial fields, including ships, and the probability that performance breakthroughs could come from local optimization methods is likely to be small. These open issues are motivating the development of high-performance, global, multiobjective optimization algorithms in association with advanced CFD methods.

Acknowledgments. This work was supported by the U.S. Office of Naval Research under grants 000140210489, 000140210304, 00140210256, 000140510616, and 000140510723 through Dr. P. Purtell and Dr. H. Narita. The authors would like to thank Dr. Angelo Olivieri for the experimental data. EFC and DP thank Dr. Di Mascio for the use of MGShip. Finally, YT expresses his appreciation to Dr. R. Wilson for his kind support for the use of CFDSHIP-Iowa version 3.02.

References

- Campana EF, Peri D, Tahara Y, et al (2006) Shape optimization in ship hydrodynamics using computational fluid dynamics. Comput Meth Appl Mech Eng 196:634–651
- Tahara Y, Paterson E, Stern F, et al (2000) Flow- and wavefield optimization of surface combatants using CFD-based optimization methods. 23rd Symposium on Naval Hydrodynamics, September 17–22, Val de Ruil (available as CD-ROM)
- Tahara Y, Sugimoto S, Murayama S, et al (2003) Development of CAD/CFD/optimizer-integrated hull-form design system. J Kansai Soc Naval Archit 240:29–36
- Tahara Y, Stern F, Himeno Y (2004) CFD-based optimization of a surface combatant. J Ship Res 48:273–287
- Tahara Y, Tohyama S, Katsui T (2006) CFD-based multiobjective optimization method for ship design. Int J Numer Meth Fluid 52:449–527
- Peri D, Campana EF, Di Mascio A (2001) Development of CFD-based design optimization architecture. 1st MIT Conference on Fluid and Solid Mechanics, Cambridge (MA), USA, June 12–14, 2001
- Peri D, Campana EF (2003) High-fidelity models in the multidisciplinary optimization of a frigate ship. 2nd MIT Conference on Fluid and Solid Mechanics, Cambridge (MA), USA, June 17–20, 2003
- Peri D, Campana EF (2003) Multidisciplinary design optimization of a naval surface combatant. J Ship Res 41: 1–12
- Peri D, Campana EF (2003) High-fidelity models in simulation-based design. 8th International Conference on Numerical Ship Hydrodynamics, Busa, South Korea, September 22–25, 2003
- Peri D, Campana EF (2005) High-fidelity models and multiobjective global optimization algorithms in simulation-based design. J Ship Res 49:159–175
- Hino T, Kodama Y, Hirata N (1998) Hydrodynamic shape optimization of ship hull forms using CFD. 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, May 25–27, Osaka, pp 533–641
- Minami Y, Hinatsu M (2002) Multi-objective optimization of ship hull form design by response surface methodology. 24th Symposium on Naval Hydrodynamics, July 8–13, Fukuoka (available on CD-ROM)

- Newman JC III, Pankajakshan R, Whitfield DL, et al (2002) Computational design optimization using RANS. 24th Symposium on Naval Hydrodynamics, July 8–13, Fukuoka (available on CD-ROM)
- Duvigneau R, Visonneau M, Deng GB (2003) On the role played by turbulence closures in hull shape optimization at model and full scale. J Mar Sci Technol 8:11–25
- Brizzolara S (2004) Parametric optimization of SWAT-hull forms by a viscous-inviscid free surface method driven by a different evolution algorithm. 25th Symposium on Naval Hydrodynamics, August 8–13, St. John's (available on CD-ROM)
- 16. Birk L, Harries S (eds) (2003) OPTIMISTIC—optimization in marine design. Mensch and Buch, Berlin
- 17. Miettinen KM (1999) Nonlinear multiobjective optimization Kluwer, Boston
- Wilson R, Paterson E, Stern F (2000) Verification and validation for RANS simulation of a naval combatant. In: Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics. Chalmers University of Technology, Gothenburg, Sweden, September 14–16, 2000
- Paterson EG, Wilson RV, Stern F (2003) General-purpose parallel unsteady RANS ship hydrodynamics code: CFDSHIP-Iowa. IIHR Report No. 432, IIHR—Hydroscience and Engineering, The University of Iowa, Iowa City
- 20. Di Mascio A, Broglia R, Favini B (2000) A secondorder Godunov-type scheme for naval hydrodynamics. In: Toro EF (ed) Godunov methods: theory and application. Kluwer, Singapore
- Larsson L, Stern F, Bertram V (2003) Benchmarking of computational fluid dynamics for ship flows: the Gothenburg 2000 workshop. J Ship Res 47:63–81
- 22. Stern F, Wilson RV, Coleman HW, et al (2001) Comprehensive approach to verification and validation of CFD simulations—Part 1: methodology and procedures. J Fluid Eng 123:793–802
- Wilson RV, Stern F, Coleman HW, et al (2001) Comprehensive approach to verification and validation of CFD simulations—Part 2: application for RANS simulation of a cargo/container ship. J Fluid Eng 123:803–810
- Coleman HW, Stern F (1997) Uncertainties in CFD code validation. ASME J Fluid Eng 119:795–803
- 25. Holland J (1975) Adaptation in natural and artificial systems. University of Michigan Press, Ann Arbor
- 26. Davis L (1990) Handbook of genetic algorithms. Van Nostrand Reinhold, New York
- 27. Deb K (2001) Multiobjective optimization using evolutionary algorithms. Wiley, New York
- Tahara Y, Katsui T, Kawasaki M, et al (2004) Development of large-scale high- performance CFD coding method for PCcluster parallel computing environment—1st report: setup and initial evaluation of coding environment with MPI protocol. J Kansai Soc Naval Archit 241:47–58
- Campana EF, Peri D, Pinto A, et al (2005) A comparison of global optimization methods with application to ship design. 5th Osaka Colloquium on Advanced Research on Ship Viscous Flow and Hull Form Design by EFD and CFD Approaches, March 14–15, Osaka, pp 164–173
- Solomatine DP (1999) Two strategies of adaptive cluster covering with descent and their comparison to other algorithms. J Global Optim 14(1):55–78
- Hedayat AS, Sloane NJA, Stufken J (1999) Orthogonal arrays: theory and applications. Springer Series in Statistics, Springer-Verlag, Berlin
- 32. Statnikov RB, Matusov JB (1995) Multicriteria optimization and engineering. Chapman and Hall, New York

- Törn AA, Žilinskas A (1989) Global optimization. Springer-Verlag, London
- Sederberg T, Parry S (1986) Free-form deformation of solid geometric models. Comput Graph 20:151–160
- Lampariello F, Sciandrone M (2001) Efficient training of RBF neural networks for pattern recognition. IEEE Trans Neural Netw 12(5):1235–1242
- 36. Meyers WG, Applebee TR, Baitis AE (1981) User's manual for the standard ship motion program, SMP. Software Documentation, DTNSRDC/SPD-0936-01 Technical Report, Bethesda, ML, USA
- 37. Newman JN (1977) Marine hydrodynamics. MIT Press, Cambridge
- Stern F, Longo J, Penna R, et al (2000) International collaboration on benchmark CFD validation data for surface combatant DTMB model 5415. 23rd Symposium on Naval Hydrodynamics, September 17–22, Val de Ruil (available on CD-ROM)

- 39. Wilson R, Carrica P, Stern F (2005) RANS simulation of a container ship using a single-phase level set method with overset grids. CFD Workshop Tokyo 2005, March 9–11, Tokyo, pp 546–551
- 40. Tahara Y, Wilson R, Carica P, et al (2006) RANS simulation of a container ship using a single-phase level set method with overset grids and prognosis for extension to self-propulsion simulator. J Mar Sci Technol 11:209–228
- Broglia R, Muscari R, Di Mascio A (2005) Computations of free surface turbulent flows around ship hulls by a RANS solver. CFD Workshop Tokyo 2005, March 9–11, Tokyo, pp 498–503
- 42. Campana EF, Peri D, Tahara Y, et al (2006) Simulation-based design of fast multihull ships. 26th Symposium on Naval Hydrodynamics, September 17–22, Rome (available on CD-ROM)

ORIGINAL ARTICLE

Single- and multiobjective design optimization of a fast multihull ship: numerical and experimental results

Yusuke Tahara · Daniele Peri · Emilio Fortunato Campana · Frederick Stern

Received: 19 April 2011/Accepted: 13 July 2011/Published online: 23 August 2011 © JASNAOE 2011

Abstract Numerical optimization of the initial design of a fast catamaran (high-speed sealift research model B, HSSL-B) has been carried out through a simulation-based design (SBD) framework, based on an advanced free-surface unsteady Reynolds-averaged Navier-Stokes (URANS) solver and a potential flow solver, and global optimization (GO) algorithms. The potential flow computational fluid dynamics (CFD) SBD was used to guide the more expensive URANS CFD SBD. The fluid-dynamic analysis of the flow past the catamaran proved that the use of the URANS solver was fundamental in dealing with the multihull interference problem. In the case investigated, the separation distance was small and the viscous flow quite distorted by the proximity of the hulls, so that only viscous solvers could correctly capture the flow details. Sinkage and trim effects, due to the high speed range and again to the small separation distance investigated, are also relevant. The initial HSSL-B geometry and three optimization problems, including single- and multiobjective optimization problems, proposed by designers from Bath Iron Works, were successfully optimized/solved, and finally an experimental campaign was carried out to validate the optimal design. A new verification and validation methodology for assessing uncertainties and errors in simulation-based optimization

Y. Tahara (🖂)

Fluids Engineering Department, National Maritime Research Institute, 6-38-1 Shinkawa, Mitaka, Tokyo 181-0004, Japan e-mail: tahara@nmri.go.jp

D. Peri · E. F. Campana INSEAN-CNR, Italian Ship Model Basin, Rome, Italy

F. Stern IIHR-Hydroscience and Engineering, University of Iowa, Iowa City, USA was used based on the trends, i.e., the differences between the numerically predicted improvement of the objective function and the actual improvement measured in a dedicated experimental campaign, including consideration of numerical and experimental uncertainties. Finally, the success of the optimization processes was confirmed by the experimental measurements, and trends for total resistance, sinkage, and trim between the original and optimal designs were numerically and experimentally verified and validated.

Keywords Fast multihull ship · Simulation-based design · CFD-based single- and multiobjective optimization · Derivative-free optimization algorithm · Total resistance minimization · Seakeeping merit function minimization · Verification and validation

List of symbols

$C_{\rm PH}, C_{\rm PP}$	Hydrostatic and piezometric
	pressure components of the
	pressure resistance coefficient
	$C_{\rm P}$, respectively
$D_{\rm P}, D_{\rm O}$	Experimental value of the parent
	and optimized designs
E_{Δ}	Difference between the measured
	and expected improvements
f	Fitness function, wave frequency
$Fr = U/\sqrt{gL_{\rm PP}}$	Froude number
g	Gravitational acceleration
KM	Metacenter height from keel
$L_{\rm PP}, L$	Ship length
$L_{\rm CB}$	Center of buoyancy
т	Population size – 1
Ν	Number of multiobjective
	functions

$\vec{F} = (F_1, F_2, \ldots, F_N)^{\mathrm{T}}$	Multiobjective functions
M	Number of design variables
n	Number of blocks
h, g	Equality and inequality constraint
	functions
$\vec{P}_1, \vec{P}_1, \vec{P}_2, \vec{P}_3$	Position vectors of hull surface
- ,- 1,- 2,- 5	points
p, q	Number of equality and inequality
1 / 1	constraint functions
$Re = UL_{PP}/v$	Revnolds number
$R_{\rm P}$	Pareto ranking
R _T	Total resistance
R ^{prf}	Total resistance of original design
S	Separation distance
S(f)	Jonswap spectrum for wave
5())	frequency f
\$/I	Separation ratio
S D	Simulation value and date value
S, D S- S-	Numerical simulation value of the
5p, 50	parent and optimized designs
\rightarrow (\rightarrow)	Valocity components, normalized
u(x)	by ship speed U
	Shin speed U
U, V_1, V_2, V_3	Unaartaintu
$U_{\text{SNP}}, U_{\text{SNO}}, \text{etc.}$	Design variables
$x = (x_1, x_2, \dots, x_M)^T$	Design variables
$x_i^{\mathrm{u}}, x_i^{\mathrm{u}}$	Upper and lower bounds of design
	variables
X, Y, Z	Nondimensional Cartesian
	coordinates, normalized by ship
	length $L_{\rm PP}$
<i>z</i> _B	Vertical acceleration at bridge
	control point
<i>z</i> _D	Vertical velocity at flight deck
	control point
α , $T_{\rm p}$, $H_{1/3}$, σ , γ	Parameters in Jonswap spectrum
_	S(f)
α, β	Design variables
Δ_i	Total displacement for speed <i>i</i>
$\Delta_{\rm S}, \Delta_{\rm D}$	Differences in value between the
	parent and optimized designs
Φ	Lamb scalar
χ	Subset χ of the <i>M</i> -dimensional
	real space \Re^M
v	Kinematic viscosity

1 Introduction

The focus of the present paper is on the use of simulationbased design (SBD) in ship design, demonstrated in a real industrial application, i.e., the design optimization of a fast catamaran, representative of present high-speed ship design trends. The base design is the HSSL-B (high-speed sealift research model B, displacement at draft 6.50 m of 12000 tons, length 170 m, beam 40 m), selected for international collaboration by ship designers to match the requirements posed by the Navy for a high-speed displacement ship with limited length and draft. Interaction among SBD developers and ship designers has led to the definition of a number of optimization problems, aimed at performance enhancement of the catamaran. Special emphasis was devoted to the selection of real geometrical constraints. The present work is connected with the Office of Naval Research HSSL project for URANS CFD-based design optimization [1, 2].

The HSSL-B has been initially developed in two versions: a catamaran and a trimaran, and later, the former was selected for the initial design optimization (Figs. 1, 2). The complexity of the design problem with the inherent difficulty of dealing simultaneously with a growing number of design goals and constraints raised the interest of ship designers in the use of a SBD framework. As a result of close interaction among code developers, in the framework of their respective Naval International Cooperative Opportunities in Science and Technology Program (NI-COP) projects, involving the National Maritime Research Institute (NMRI), IIHR-Hydroscience and Engineering of the University of Iowa, and the Italian Ship Model Basin (INSEAN), and ship designers from Bath Iron Works, a number of optimization problems, aimed at enhancement of hydrodynamic performance, were discussed and



Fig. 1 Initial HSSL-B design in the catamaran version. The separation ratio between the hulls is S/L = 0.16



Fig. 2 Side view of the HSSL-B; the flight deck and bridge locations, where the functional constraints on vertical acceleration and vertical velocity are enforced, are shown

elaborated. Special emphasis was devoted to selection of geometrical and performance constraints too. It is indeed common experience that simple approaches combining CFD solvers and optimization algorithms often lack robustness, can be very inefficient, or eventually produce meaningless solutions from the standpoint of producibility. The collaboration helped in overcoming challenges arising from the increasing complexity of the real application. This was the necessary background for the development of reliable SBD and forms the basis of the present work.

Regarding the specific design problem, it was found that, notwithstanding the large body of literature dealing with multihulls, papers devoted to analysis of viscous flow past a catamaran or trimaran with Reynolds-averaged Navier-Stokes (RANS) simulations are very limited. Most of the previous works indeed adopt potential flow models (e.g., [3]) to predict the so-called interference effect and to select the optimal separation distance between the demihulls. (See also the discussion on interference effects by [4].) On the contrary, use of state-of-the-art unsteady Reynolds-averaged Navier-Stokes (URANS) solvers [2] has instead revealed the extension of the flow distortion induced by the proximity of the hulls. Compared with the monohull (demihull) case, distorted streamlines, altered pressure distributions from bow to stern, zones of flow separation, and vortex shedding have been easily detected (Fig. 3). Each of these effects, which could not be captured by inviscid flow solvers, has the potential to alter the resistance characteristics of the multihull. Another effect known to be relevant is produced by the sinkage and trim [2]. Due to the high speed range and the proximity between



Fig. 3 BIW-A trimaran vortical structures [2]

the demihulls, the sinkage and trim proved to be relevant in the estimation of the resistance, whose prediction was fundamental to guide the optimizer in the right direction.

The optimization scheme plays another major role in the present SBD framework. Initially, we introduced local optimization schemes into our SBD frameworks, which were exercised through single- and multiobjective optimization to redesign the DTMB 5415 model [5, 6]. Experimental validations of the results as well as the scheme were also performed, and the success of optimization was confirmed. However, the limitation of the local optimization scheme was obvious; i.e., typical industrial cases translate into nonlinear, multimodal, nonconvex problems, and the true optimal solution is difficult to find, hidden in a region with many local minima. In this scenario, local optimizers are not able to overcome local minima, and cannot deal with nonconnected design spaces, jumping from one feasible region to another. Multistart methods, based on a local algorithm starting its repeated searches from a cloud of randomly distributed initial points, appear to be an easyto-implement solution with less chance of becoming trapped in local optima, but with poor or no insight into the problem. True global optimization (GO) methods appear to be the right way to address these difficulties (e.g., [7]). With this motivation, GO methods based on a derivativefree algorithm together with parallel computing architecture were developed in the present work.

The above-mentioned URANS solver and GO method together with a geometry modeling method were integrated to realize the SBD framework, and the single- and multiobjective optimization problems proposed by the designers were solved. In addition, a potential flow CFD-based SBD (i.e., PF CFD SBD) framework was also composed. Since the PF CFD SBD is less expensive, a greater number of design variables was used to explore even somewhat drastic redesigns. Then, the results were used to guide more expensive URANS CFD-based optimization (i.e., URANS CFD SBD), which uses fewer design variables. In practice, this approach is the variable fidelity/variable physics approach in the ship hydrodynamic research area, although at present, the interaction is still indirect, requiring future work to develop it in a more systematic manner, as described later.

The initial HSSL-B geometry and three optimization problems, including single- and multiobjective optimization problems, proposed by designers from Bath Iron Works, were successfully optimized/solved, and finally an experimental campaign was carried out to validate the optimal design. The experimental data were then used in a new verification and validation (V&V) procedure based on analysis of the trend of the objective function to be minimized. Indeed, due to the inherent uncertainties in simulations and experiments, the success of an optimization process is more appropriately based on trends than absolute values.

2 Multihull design and needs for high-fidelity analysis method

A common practice in development of multihulls is to use monohull forms for the individual demihulls. Most knowledge of ship resistance is indeed derived from development and testing of monohulls at model scale with validation at full scale. The approach used is first to measure total model resistance, then to estimate the frictional resistance component (assumed to scale with the Reynolds number, Re) using a standard friction line, and finally to derive a residuary resistance component (assumed to scale with the Froude number, Fr) by subtracting the frictional resistance from the total resistance. Although there is some influence of other considerations such as seakeeping and powering, the resulting hull forms primarily reflect efforts to reduce resistance. Typically, these hull forms have streamlined distributions of underwater volume that are symmetric about the hull centerplane, being designed to produce relatively small disturbances to the otherwise uniform flow of water past the body.

Hence, especially in the development of catamarans, ship designers are then tempted to use monohull forms for the individual demihulls. However, the flow about two (or more) of these monohull-like forms is not symmetric about the centerplane of all of the demihulls. Depending mainly on the proximity of one hull to the other, the flow is indeed distorted by their mutual presence. Consequently, the path of a water particle about a demihull differs substantially from that encountered in passing a hull in isolation. This difference in flow typically (but not always) results in an increase of the resistance.

The resistance of a multihull composed by n demihulls is substantially greater than n times the resistance of one hull in isolation. This added resistance is generally considered to be part of the residuary resistance and is attributed to the interference effect, a term that relates the demihull (or monohull) drag to the multihull drag. The added drag is principally due to: (a) the interaction of the wave trains from the individual demihulls (i.e., wavemaking interference), and (b) the changes in the viscous resistance of each demihull due to distortions in the flow caused by the presence of the other demihulls. Traditional approaches to hull form optimization are focused on wave interference. However, pressures induced on each hull by the presence of the other demihull(s) significantly distort the flow over each demihull in ways that affect viscous resistance as well as wavemaking.

A considerable amount of towing tank work has been done to measure multihull interference drag associated with variations in demihull longitudinal and transverse positions. However, little experimental data exists to provide insight into interference effects between hulls in the absence of the free surface. Fortunately, a dataset exists for a catamaran hull with systematic variation in hull separation for both a ship model in a towing tank [8] and a double model of the hull in a wind tunnel [9]. This set of data shows that the altered flow due to the proximity of the two hulls results in significant interference effects in both the wind tunnel and the towing tank. In summary, there are significant interference effects between demihulls in the absence of a free surface, which are independent of wavemaking interactions. If factors such as wind tunnel blockage effects, model mounting effects, etc. can be discounted, the inescapable conclusion is that there is significant interference of a purely viscous nature between two scale-model hulls in close proximity.

Towing tank tests combine viscous interference effects with wavemaking interference effects. A comparison of Armstrong's wind tunnel data with Molland's towing tank data is shown in Fig. 4. The upper three curves in the figure show (going down from the top) total towing tank drag, total towing tank drag minus wave profile drag, and total wind tunnel viscous drag. In the absence of any interference, the difference between the total towing tank drag and the total viscous drag from the wind tunnel would equal the wave profile drag. However, the figure clearly shows that there is drag interference in addition to the wavemaking interference inherent in the wave profile drag.

This viscous-inviscid interference is most significant in the vicinity of the hump in the towing tank drag curve at Fr = 0.5, near the HSSL design point. The data in the figure show this difference to be in the 10–25% range. While other data from Armstrong show the viscousinviscid interference to be dependent on hull separation, he did not test for hulls as close as the HSSL catamarans (S/L = 0.16). While it is tempting to hope that this added interference drag involves only a simple correction to the viscous drag, the variation of the interference effect with speed indicates a significant Fr effect. As already stated, use of RANS solvers is then fundamental in dealing with



Fig. 4 Reproduced from [9]

the multihull interference problem, and the small separation distance of the HSSL-B enhances the flow distortion.

3 Single- and multiobjective optimization problem proposed by designers

Our goal is to redesign the HSSL-B catamaran by using SBD frameworks. For the optimization problem, three different design cases, of increasing difficulty, were formulated. In Table 1 the objective functions along with the geometrical and functional constraints for the 3 cases are shown. Case 1 is a relatively simple, single-objective function problem. The difficulty comes from the flow complexity and the relatively high speed range. Case 2 is a weighted approach (with equal weights of 1/3) to a multipoint design. The three different displacements correspond to the full loaded condition down to the return from the mission condition, to which correspond three different speeds. The length-based Fr of the average speed is 0.541. The other two Fr represent the departure (initial) and arrival (final) speeds at constant power and varying displacement as the ship changes from fully loaded

at departure to nearly burned out at arrival. Case 3 is a multiobjective problem with resistance and seakeeping criteria, involving the root-mean-square (RMS) vertical acceleration at the bridge and the RMS vertical velocity at the flight deck (Fig. 2).

The PF CFD SBD and URANS CFD SBD solve equally formulated optimization problems. Shape design optimization is typically formulated in the framework of a nonlinear programming (NLP) problem. For a general expression of an N-objective-function optimization problem in ship hydrodynamics, the mathematical formulation assembles all the design variables $x_1, x_2, ..., x_M$ in a vector $\vec{x} = (x_1, x_2, \dots, x_M)^{\mathrm{T}}$ belonging to a subset χ of the *M*-dimensional real space \Re^M , i.e., $\vec{x} \in \chi \subseteq \Re^M$ (upper x_i^u and lower x_i^{l} bounds are typical enforced on the design variables). The objective of the optimization $\vec{F} =$ $(F_1, F_2, \ldots, F_N)^{\mathrm{T}}$ and the equality and inequality constraints h, g are functions of the design variables \overline{x} and of the state of the system $\vec{u}(\vec{x})$. A general form for constrained NLP problems is then to find the particular vector \tilde{x} in the subset γ which solves the following:

Table 1 Problem description: objective functions, and functional and geometrical constraints

Test #ID	Objective function	Geometrical constraints	Functional constraints
1.	Single-objective (one-speed) problem: Minimize $R_{\rm T}$ at $Fr = 0.541$ for the ship free to sink and trim	 a. Max. overall length (170.7 m) and max. beam (40 m) b. Draft ≤ 6.5 m c. Total displacement 10785 t d. 0.3 ≤ L_{CB}/L ≤ 0.7 e. KM_{T_Original} ≤ KM_{T_Optimal} KM_{L_Original} ≤ KM_{L_Optimal} (individual hull waterplane area ≥ 150 m²) 	None
		f. Immersed transom area: still-water transom wetting is equal to one waterjet semidiameter (1.59 m) in the vertical direction and two (6.36 m) in the transverse direction	
		g. More than 1 m above the keel and from $L/2$ to stern, the distance between port and starboard shells is ≥ 1 m	
2.	Single-objective (three-speed) problem: with $\alpha_i = (1/3, 1/3, 1/3), Fr_i = (0.460, 0.541, 0.622),$ and $\Delta_i = (12000, 10785, 9570)$ t, minimize $F = \sum_{i=1}^{3} \alpha_i \frac{R_{\rm T}(Fr_i)}{V_i^2}$	As in problem #1 but for c. Total displacement Δ_i depending on the speed	None
3.	for the ship free to sink and trim <i>Multiobjective (resistance and seakeeping) problem:</i> with <i>B</i> (bridge) = (128.025, 0, 15) m, <i>D</i> (flight deck) = (21.3375, 0, 5) m, sea state 5 (head seas), $\ddot{z}_{\rm B}$ the RMS of the vertical acceleration at the bridge, $\dot{z}_{\rm D}$ the RMS of the vertical velocity at the flight deck, and VCB = 4.193 m, minimize $F_1 = R_{\rm T}(0.541)$, $F_2 = 0.5 \frac{\ddot{z}_{\rm B}}{0.2g} + 0.5 \frac{\dot{z}_{\rm D}}{1.0}$ for forward speed of $Fr = 0.541$	As in problem #1	a. $\frac{R_{\rm T}(0.460)}{R_{\rm T}^{\rm phf}(0.460)} \le 1.0$ b. $\frac{R_{\rm T}(0.622)}{R_{\rm T}^{\rm phf}(0.622)} \le 1.0$ c. $\ddot{z}_{\rm B} \le 0.2g$ d. $\dot{z}_{\rm D} \le 1.0$ m/s

Deringer

$$\operatorname{Min:} \begin{cases} [F_1(\vec{x}, \vec{u}(\vec{x}); Re, Fr)]_{Re,Fr} \\ [F_2(\vec{x}, \vec{u}(\vec{x}); Re, Fr)]_{Re,Fr}, & x \in \chi \subseteq \Re^M \\ \vdots \\ [F_N(\vec{x}, \vec{u}(\vec{x}); Re, Fr)]_{Re,Fr} \end{cases}$$
(1)

Subject to : $h_j(\overline{x}) = 0 \ (j = 1, ..., p)$ $g_j(\overline{x}) \le 0 \ (j = 1, ..., q)$

$$g_j(\vec{x}) \le 0 \ (j = 1, ..., q)$$

 $x_i^1 \le x_i \le x_i^u \ (i = 1, ..., M).$

The aforementioned case 3 problem belongs to the above formulation; on the other hand, for a single-objective optimization problem, i.e., case 1 and case 2 problems, the above reduces to

Min:
$$[F(\vec{x}, \vec{u}(\vec{x}); Re, Fr)]_{Re, Fr}, \quad x \in \chi \subseteq \Re^M$$
 (2)

Subject to :
$$h_j(\vec{x}) = 0 \ (j = 1, ..., p)$$

 $g_j(\vec{x}) \le 0 \ (j = 1, ..., q)$
 $x_i^l \le x_i \le x_i^u \ (i = 1, ..., M).$

The solution of the above problems typically requires the use of some numerical tool—the first constitutive element of the SBD frameworks—to solve the system $\mathbf{A}(\vec{x}, \vec{u}(\vec{x})) = 0$ and evaluate the current design \vec{x} , obtaining information on the constraints too. If the function used to define the optimization problem is of fluid-dynamic nature, as in our case, the step requires the evaluation of the design \vec{x} via a CFD solver, a process which is itself computationally intensive. Within a standard nonlinear optimization algorithm—the second fundamental element of a SBD—the solution of these differential equations is required for each iteration of the algorithm.

In addition to these two elements, a third one is necessary: a geometry modeling method to provide a link between the design variables and the body shape. When the analysis tools are based on the solution of a partial differential equation (PDE) on some volume grid around a complex geometry, this task is not a trivial one and often requires some attention. The flexibility of this element may greatly affect the freedom of the optimizer to explore the design space.

The aforementioned three problems, i.e., case 1 through case 3, were successfully solved using the present two SBD frameworks. Figure 5 shows geometry comparison among the original and three optimal designs, and Tables 2 and 3 show comparison of main particulars of the designs, and optimization results for total resistance, objective functions, sinkage, and trim. In the following, details of the two alternative SBD frameworks are described and the results are discussed.

4 Potential flow CFD-based optimization

4.1 Shape parameterization

The method adopted to parameterize the shape is the freeform deformation (FFD) technique. In FFD, the object to be deformed is virtually embedded into a parallelepiped (Fig. 6a). This scheme was also adopted in the authors' earlier work [5, 6]. One of the advances in the present work was to develop a general parameterization approach, able to deal with a large number of variables, say 50. The choice of these 50 variables is illustrated in Fig. 6.

x variations (Fig. 6b): in each of the four internal sections, all the points are grouped together, resulting in 1 variable \times 4 sections = 4 variables;

y variations (Fig. 6c): points are clustered together into groups of four vertices. Besides the four sections visible in Fig. 6b, one more section is added at the extreme fore position, to control the bow shape. Hence we have 6 variables \times 5 sections + 4 variables on the bulb = 34 variables;

z variations: in each of the four internal sections (Fig. 6b) and in the two extreme aft and fore sections, the points are clustered into two groups (Fig. 6d), resulting in 2 variables \times 6 sections = 12 variables.

This strategy allows for complete modification of the hull surface by utilizing 50 design variables.

4.2 Optimization algorithm

The optimizer is a derivative-free hybrid approach, combining an evolutionary algorithm (particle swarm optimization) with a direct search technique (diagonal rectangular algorithm for global optimization, DRAGO), details of which are described in [7, 10, 11], respectively. The hybrid PSO-DRAGO strategy tries to get the best from both algorithms. While PSO tends to concentrate the optimum search in the most promising region of the design space, DRAGO, on the contrary, tends to check all the unexplored, large intervals before proceeding to a next interval subdivision. Every other iteration, the method applied switches between these two, offering a blending strategy, which was found to be effective [12]. A flowchart of the PSO-DRAGO approach is shown in Fig. 7.

4.3 CFD: potential flow solver

The wave resistance prediction (WARP) potential flow solver adopted for the analysis in the present version of SBD is based on the quite classical linearization of the freesurface conditions around the calm water level. Details of equations, numerical implementation, and validation of the

-448-



a Comparison of bodyplan. PF CFD-based optimization. Contour interval d=0.05.



c Comparison of load waterplane. PF CFD-based optimization.

Fig. 5 Comparison of geometry among the original and three optimal designs. Case 3 NO 203 and case 3 ID-556 are F_2 -minimum designs on Pareto set for PF CFD SBD and URANS CFD SBD, respectively.



b Comparison of bodyplan for URANS CFD-based optimization. Contour interval d=0.05.



d Comparison of load waterplane. URANS CFD-based optimization.

Case 3 ID-151 is a $F_1\!-\!F_2$ compromise design on Pareto set for URANS CFD SBD

Table 2 Comparison of main particulars among the original		Optimal model					
and optimal designs	designs		Case 2	Case 2 (%)		Case 3 (NO 203) (%)	
	PF CFD-based optimization						
	$L_{\rm PP}{}^{\rm a}$	100.0	100.0		100.0		
	Displacement ^a	100.6	100.5		100.6		
	Wetted surface area ^a	107.3	107.9		107.3		
	$\Delta L_{\rm CB}{}^{\rm b}$	+8.6	+8.6		+8.6		
		Optimal model					
		Case 1 (%)	Case 2 (%)	Case 3 (ID-556) (%)		Case 3 (ID-151) (%)	
	URANS-based optimization						
	$L_{\rm PP}{}^{\rm a}$	100.0	100.0	100.0		100.0	
a ~ · · 1	Displacement ^a	100.0	100.0	100.0		100.0	
" % original	Wetted surface area ^a	104.6	104.7	104.6		104.6	
$\sim \% L_{\rm PP}$ original, positive backward	$\Delta L_{\rm CB}{}^{\rm b}$	+6.4	+6.4	+6.4		+6.4	

Table 3 Comparison of optimization results for total resistance, objective functions, sinkage, and trim

Optimal model	$R_{\rm T}/V^{2~a}$			F^{a}	F_1^{a}	F_2^{a}	Sinkage ^a	Trim ^a
	Fr = 0.460 $Fr = 0.541$		Fr = 0.622		Fr = 0.541		Fr = 0.541	
PF CFD-based optimization	n							
Case 1 (%)	_	60.7	_	60.7	-	-	76.8	69.2
Case 2 (%)	78.8	72.2	56.0	69.0	-	-	60.7	64.1
Case 3 (NO 203) (%)	_	97.9	_	-	97.9	97.0	_	-
URANS CFD-based optimi	ization							
Case 1 (%)	_	90.3	_	90.3	-	-	49.5	72.6
Case 2 (%)	95.8	90.4	81.0	91.7	-	-	49.9	72.9
Case 3 (ID-556) (%)	-	91.4	-	-	91.4	21.5	49.9	72.3
Case 3 (ID-151) (%)	-	90.7	-	-	90.7	59.5	49.6	72.6

^a % original

numerical solver are given in [13]. WARP, developed at INSEAN in the late 1980s and continuously improved since then, is routinely used at INSEAN for resistance evaluations, and experience in multihull predictions and on the effects of demihull spacing has already been accumulated (e.g., [14]).

4.4 Numerical optimization results

The above-described geometry modeling scheme, optimization scheme, and CFD method compose a PF CFD SBD to demonstrate HSSL-B optimization test cases. The optimizations were performed at INSEAN by using a PC cluster of 24 Intel Xeon (64 bit, 3.2 GHz). As stated before, attention was focused on the increase of the number of design variables up to 50. It has to be remembered also that the computations were performed with the catamaran

free to sink and trim, with a consequent increase of central processing unit (CPU) time.

Case 1 results: In the solution of case 1, the initial design of experiments (DOE) sampling of the design space required 160 trial designs. These data served to build a kriging metamodel type adopted to solve the optimization problem. The CPU time required for the solution of the kriging, i.e., to get an approximate value of the objective function, become in that way negligible. For that reason, the CPU time required to get an optimum value using either the PSO or the DRAGO algorithm becomes almost insignificant compared with the training phase or the numerical verification of the discovered optimum via the potential flow solver. A summary of the CPU time for each step of the algorithms in given in Table 4. The optimized geometry (Fig. 5a, c) shows a clear improvement of the wave pattern both in between the demihulls and at the stern

Fig. 6 Parameterization via the FFD approach: a demihull embedded in the deforming parallelepiped; b control points governing the *x* changes; c and d control points governing *y* and *z* deformation, respectively. *Same colors* indicate grouped control points. For PF CFDbased optimization





Fig. 7 Flowchart of the hybrid PSO-DRAGO scheme. The algorithm was tailored for a cluster with N processors. k is the step in the process. For PF CFD-based optimization

(Fig. 8). Since the present potential flow solver predicts only wavemaking resistance, total resistance is estimated by including frictional resistance given by the ITTC57 formula. The estimated total resistance is significantly reduced (Table 3). The demihull is very slender, hence increasing the demihull spacing while preserving the maximum beam. To keep the displacement constant, the submerged volume near the stern region is increased. Sinkage and trim values were reduced too, with respect to the original design (Table 3).

Case 2 results: In the solution of case 2, the same number of variables was used. Results are reported in Figs. 5 and 9. In interpreting the results it has to be remembered that the potential flow solver adopted is based on a linearization around the undisturbed water level, i.e., the flow is computed only around the underwater part while

the part above the static waterplane has no influence on the optimization process and can therefore be reshaped. Again, the SBD finds the same correct trend; i.e., it tries to make the demihull more slender with respect to the original HSSL-B to increase the demihull spacing while preserving the maximum beam. To satisfy the displacement constraint, the solution increases the submerged volume in the stern region.

The external side is flat and straight. The most evident difference with respect to both the original hull and the case 1 optimum is that the demihulls no longer have a straight keel line. Indeed, it can be seen that the shape is slightly bent at the bow. Total resistance is significantly decreased as a result of the large decrease of wave resistance (Table 3). A comparison between case 1 and case 2 is possible by looking at Table 3. Quite obviously, for the medium (design) speed, the solution of case 1 performs better than the solution of case 2, since the latter represents a trade-off among the performances at three different speeds. Again, sinkage and trim values were reduced with respect to the original design (Table 3).

Case 3 results: The same number of variables as for the previous cases was used. For this problem, seakeeping performance is evaluated by using FreDOM, which is a frequency-domain, Rankine-source-type panel method developed at INSEAN [15]. For free surface and hull surface, 4000 and 1000 panels were used, respectively. Pitching and heaving motions are considered; hence, only the starboard-side domain is considered. As specified in Table 1, sea state 5 (head sea, $T_p = 9.7$ s, $H_{1/3} = 3.25$ m)

Table 4 CPU time summaryfor the PSO/DRAGO	Step	CPU time (h)
optimization approach with	DOE training phase (160 designs)	1120
approximations	PSO-DRAGO solutions using the metamodel	~ 0
approximations	Potential flow analysis of the suboptimal designs	140
	2.5 iterations = total CPU time on a 20-node cluster	3.8 days

Fig. 8 Results for case 1. Perspective view of the original (*left*) and optimized (*right*) catamaran. Free surface is colored with wave height. For PF CFD-based optimization





Fig. 9 Results for case 2. Perspective views of the wave pattern for the 3 different speeds (Fr = 0.460, 0.541, 0.622). Free surface is colored with wave height using the same contours levels as in Fig. 8. For PF CFD-based optimization

is assumed, for which the following Jonswap spectrum is applied:

$$S(f) = \frac{\alpha H_{1/3}^2}{T_p^4 f^5} \exp\left\{-\frac{1.25}{(T_p f)^4}\right\} \gamma^{\exp\left\{-\frac{(T_p f - 1)^2}{2\sigma^2}\right\}}.$$
(3)

This is used to evaluate the RMS vertical acceleration at the bridge and the RMS vertical velocity at the flight deck. The locations of evaluation are shown in Fig. 2. The same numerical method is used for URANS CFD SBD.

Figure 10 shows the distribution of solutions, where the curve labeled "Pareto front" indicates the location of the Pareto-optimal set, all designs of which will be candidates for design trade-off between F_1 and F_2 . From the Pareto optimal set, no. 203 (F_2 -minimum design) is selected for further evaluation. Table 3 shows percentage differences of objective functions for design no. 203, where F_1 and F_2 decrease by 2.1% and 3%, respectively, compared with the



Fig. 10 Results for case 3. From the Pareto-optimal set, no. 203 is selected for further evaluation. For PF CFD-based optimization

original design. Figure 5a, c shows geometry comparison between the original and optimal designs. General geometrical features are similar to those for case 1 optimal design, but the width of the demihull is slightly increased.

5 URANS CFD-based optimization

5.1 Shape parameterization and grid deformation

The hull form was modified using two different approaches: type A, direct movement of hull surface points in particular direction, which is combined in multiple directions; and type B, hull form blending by using several basic hull forms. A similar approach is used in [16, 17], respectively. Figure 11 shows an overview of this approach. Type A modification was used for the case 1 and case 2 optimization problems. The function was defined so that important trends predicted in PF CFD to reduce wave-making resistance for case 1 and case 2 were reproduced. Six design parameters were used in conjunction with use of the simple spline function, i.e., two for inward half-body transverse modification, two for outward half-body transverse modification, and two for vertical direction modification near the stern. This strategy yields more conservative redesign than that shown in PF CFD-based optimization, i.e., allowing the keelline to be only a straight line.

Type B modification was used for case 3 optimization. The advantage of using this approach was confirmed in [17]. The number of design parameters was one fewer than the number of hulls used for the blending operation, e.g., two parameters were used for three-hull form blending as follows:



Fig. 11 Basic strategy of hull form modification, i.e., types A and B. Type A is a combination of one-directional expansion and reduction of surface. Type B is a blending or morphing technique. For URANS CFD-based optimization

Deringer

$$\vec{P} = a_1 \vec{P}_1 + a_2 \vec{P}_2 + a_3 \vec{P}_3$$
where
$$\begin{cases}
a_1 = \alpha \\
a_2 = (1 - \alpha)\beta \\
a_3 = (1 - \alpha)(1 - \beta) \\
\text{So that } a_1 + a_2 + a_3 = 1,
\end{cases}$$
(4)

where P_1 through P_3 are hull surface points for three basic designs, and $0 \le \alpha \le 1$ and $0 \le \beta \le 1$ are design variables. Once the hull form is modified, the computational grid is automatically generated by using the same algebraic scheme as used in previous work [6].

5.2 Optimization algorithm

In the present study, the previous version of the optimization module based on a binary-coded multiobjective genetic algorithm (BC-MOGA, [6]) was extended as a more capable global optimizer (GO) by introducing real-coded MOGA (RC-MOGA) and unimodal normal distribution crossover (UNDX, [18]). The advantage of RC-MOGA over BC-MOGA in engineering applications was discussed in [19]. The basic algorithm is illustrated in Fig. 12. For single-objective optimization problems (case 1 and case 2), the fitness function f is directly related to the objective function F to be minimized, i.e., $f = 1/(1 + e^{F})$. For multiobjective optimization problems (case 3), higher fitness f is given to individuals of higher Pareto ranking $R_{\rm P}$, i.e., $f = 1/R_{\rm P}$. (See [6] for more details regarding this procedure.) The drawback of algorithms of the evolutionary family, i.e., increase in computational load, is overcome by introducing a parallel computing technique, i.e., the message passing interface (MPI) protocol.

5.3 CFD: URANS solver CFDSHIP Iowa

The CFD method adopted was CFDSHIP-IOWA version 4, which is a general-purpose, multiblock, high-performance, parallel computing URANS code developed for

computational ship hydrodynamics. The URANS equations are solved using higher-order upwind finite differences, PISO, and an isotropic blended $k - \omega/k - \varepsilon$ two-equation turbulence model. The free surface is modeled using a steady and unsteady single-phase level set method to handle both complex ship geometry and complex interfacial topology due to higher *Fr*, bluff geometry, and/or large-amplitude motions and maneuvering. For more details see [20]. Overset grids are used to provide flexibility in grid generation, local grid refinement, and for bodies and/or blocks with relative motions.

5.4 Numerical optimization results

The above-described geometry modeling and griding schemes, optimization scheme, and CFD method compose a URANS CFD SBD to demonstrate HSSL-B optimization test cases. The optimization was performed at NMRI by using a PC cluster environment with 60 Intel Xeon-3070 (64 bit, 2.66 GHz). In the present optimizations, the population size of the genetic algorithm coincides with the number of MPI groups, each of which utilizes three CPUs, i.e., the flow domain in a three-block grid system is solved by using three CPUs to fully enhance the computational efficiency. Hence, the total number of CPUs used is n(m + 1) + 1 = 49 (i.e., n = 3, m + 1 = 16, where m + 1 is the population size). In the present study, a threeblock grid system is used; i.e., two blocks are used for each half inward and outward body, and one block is used for the background grid. The grids contained 273,060 points for hull blocks and 205,875 for background block, for a total of 478,935 points. Basically, the same GA system parameters are used for case 1 through case 3, i.e., crossover rate of 0.75, population size of 16, and maximum of 50 generations.

Case 1 results: among all designs generated in the 50 generations, the best design that satisfied all constraints was selected as a solution of the present problem. An optimal design that reduced the total resistance (i.e., the objective

Fig. 12 High-performance parallel computing architecture for the multiprocess CFD-based global optimization (GO) algorithm. For URANS CFDbased optimization. m + 1 is the number of population, and n is number of processors used for each CFD execution



Deringer





function) by -9.7% was obtained (Table 3). As shown in this table, sinkage and trim values were also reduced, by -50.5% and -27.4%, respectively. Figure 13 shows comparison of the wave profile between the original and optimal designs. A general trend in the optimal design is reduction of the amplitude of both inward and outward wave crests near the bow, which clearly contributes to reduction of wavemaking resistance. In addition, the location of the inward after-body wave trough is shifted backward, which causes increase of pressure in the region and also contributes to reduction of the pressure resistance. As shown in Fig. 5b, d, the demihull is more slender than in the original design, which in practice increases the demihull spacing while preserving the maximum beam. The volume distribution is moved backward and downward.

Case 2 results: as for case 1, the best design in the 50 generations was selected as the solution of the present problem. An optimal design that reduced the objective function by -8.3% was obtained, found to be nearly identical to that for case 1 (Fig. 5b, d). Consequently, as shown in Table 3, reductions of total resistance R_T , sinkage, and trim values at medium Fr (i.e., Fr = 0.541) were nearly equal to those for the case 1 optimal design, i.e., -9.6%, -50.1%, and -27.1%, respectively. It is also seen from Table 3 that reductions of total resistance at lower Fr (i.e., Fr = 0.460) and higher Fr (i.e., Fr = 0.622) were also achieved, i.e., -4.2 and -19%, respectively. As expected, the wave profile for the present optimal design is nearly identical to that for the case 1 optimal design (Fig. 13).

Figure 14 provides an explanation for why the optimal geometries for case 1 and case 2 are so similar. The figure shows a comparison of trends in the objective function F versus R_T/V_1^2 , R_T/V_2^2 , and R_T/V_3^2 for the designs generated during the present optimization, where V_1 , V_2 , and V_3 correspond to Fr = 0.460, 0.541, and 0.622, respectively. It is seen that R_T/V_1^2 , R_T/V_2^2 , and R_T/V_3^2 exhibit more or less different trends against F; on the other hand, as F becomes smaller, R_T at Fr = 0.541 (the objective function of



Fig. 14 For case 2. Objective function *F* versus R_T/V_1^2 , R_T/V_2^2 , and R_T/V_3^2 . All values are normalized by *F* of the original design. For URANS-based optimization

case 1) generally tends to be smaller. Since the same constraints are imposed for case 1 and case 2, a nearly identical design was likely to be obtained in the present case 2 optimization.

Case 3 results: for this problem, two objective functions F_1 and F_2 must be evaluated, i.e., the total resistance R_T and the seakeeping merit function, respectively, the latter being evaluated using the same numerical scheme as used for the PF CFD-based optimization, i.e., FreDOM, along with the same approach to evaluate the vertical velocity and acceleration at the flight deck and bridge for sea state 5. In the type B hull-form modification method used for the present case, three basic designs were blended to yield a new design. The three basic designs, namely B.1, B.2, and B.3 are: B.3—case 1 optimal design (i.e., R_T minimum



Fig. 15 Comparison of geometry among the original and three basic designs for blending operation (case 3), i.e., B.1, B.2, and B.3. Contour interval d = 0.05. For URANS CFD-based optimization

design), and B.2 and B.1-redesigned versions of B.3 formed by moving lower part volume distribution inward. Since design B.3 satisfied all the constraints for case 3 (Table 1), designs B.1 and B.2 were designed so as to satisfy these constraints as well. The longitudinal area distribution of B.1 and B.2 are the same as that for B.3, so that similar resistance characteristics are expected. Designs B.1 and B.2 are aimed to have different seakeeping characteristics due to change in framelines. Figure 15 shows comparison of geometry among the original and the three basic designs for the blending operation. The trends in F_1 and F_2 for those designs are shown in Fig. 16.

Finally, 800 new designs were generated by using the present RC-MOGA. Figure 16 shows the distribution of the designs, where the curve labeled "Pareto set" indicates the location of the Pareto-optimal set. All designs in the Pareto set will be candidates for design trade-off between F_1 and F_2 , since the design constraints in Table 1 are satisfied by all of the new designs in the set. In fact, this is one of the



Fig. 16 Results for case 3. From the Pareto-optimal set, MO ID-151 is selected for further evaluation. For RANS CFD-based optimization

advantages of introducing the present blending approach; i.e., if all of the basic designs satisfy the design constraints, the new design from the blending operation is more likely to satisfy the constraints as well. It is also shown that B.3 belongs to the Pareto set and still has the smallest F_1 among all the designs. From the Pareto-optimal set, a F_1-F_2 F_2 decrease by 8.6% and 78.5%, respectively.

SBD results and trends in optimal design

In the following, the focus of discussion is on comparison of optimal designs between the two SBDs and cross-evaluation of the optimal designs produced by both CFD methods. Figure 5 shows comparison of optimal designs for all case problems investigated in the present study. In addition, Tables 2 and 3 show comparison of main particulars, total resistance, objective functions as well as sinkage and trim for those designs.

Regarding reduction of F_1 , i.e., R_T , common trends for both SBD results are: (1) outward movement of volume distributions to yield a more slender demihull that increases the spacing between the two demihulls, and nonsymmetric framelines with respect to the demihull keelplane; and (2) downward movement of volume distributions near the stern in association with backward shift of L_{CB} . As shown in Table 2, the displacement of the optimal design is nearly

compromise design MO ID-151, and F_2 minimum design MO ID-556 were selected for further evaluation. Figure 5a, d, and Table 3 show comparison of geometry and objective functions, respectively. For the MO ID-556 design, F_1 and 6 Comparison of PF CFD SBD versus URANS CFD

equal to that of the original design, but the wetted surface area is increased for all optimal designs. It is also shown that PF CFD SBD tends to yield "wedge"-type designs with a clearly cut transom stern. The resultant trends in wave field are also the same between the two SBDs; i.e., the amplitude of both the inward and outward wave crest near the bow is reduced, and the inward after-body wave trough is shifted backward. In addition, sinkage and trim are significantly reduced.

For case 1, to investigate the optimal designs from both SBDs, the PF CFD SBD optimal design for this case was evaluated by using the present URANS CFD method. The same number of grids as well as overset grid structure were used. Figures 17 and 18 show comparison of URANS CFD results for wave contours and wave profiles, respectively. The aforementioned trends in wave field for reduction of $R_{\rm T}$ are seen for the two designs. However, as reported in Table 5, the URANS CFD result for $R_{\rm T}$ of the PF CFD SBD optimal design does not indicate reduction, which appears to be mainly due to increase in the hydrostatic pressure component (i.e., $C_{\rm PH}$). However, the piezometric pressure component (i.e., $C_{\rm PP}$, where $C_{\rm P} = C_{\rm PP} + C_{\rm PH}$ and $C_{\rm P}$ is the pressure reduction, which is almost of the same

magnitude as that for the URANS CFD optimal design. This is consistent with the aforementioned feature of the wave field and linearized free-surface theory used for the PF CFD; i.e., the reduction of piezometric pressure component correlates with reduction of wavemaking resistance. It appears for the present applications that fully nonlinear free-surface theory must be used for accurate prediction of $C_{\rm PH}$, and inclusion of viscous effects is also important to resolve the flow near the transom (especially for the PF CFD SBD optimal design) and associated trends in $C_{\rm PH}$.

For the case 2 optimal designs, although there are large differences between the two SBD designs, common trends in total resistance are seen (Table 3); i.e., in association with an attempt to reduce the objective function, R_T/V^2 for the given three speeds is simultaneously reduced, and the gain is smaller and larger for lower and higher *Fr*, respectively. Similarities in geometry between the two designs are related to the aforementioned trends for reduction of R_T , and especially for PF CFD SBD, the design differs from that for the previous case more significantly. On the other hand, the differences in geometry between the two SBDs for the present case are mainly attributed to frameline shapes, which are apparently due to differences in the way the design is modified; for example,



Fig. 18 Comparison of wave profiles between PF CFD and URANS CFD-based optimization designs for case 1. Both evaluated by URANS CFD



Table 5 Comparison of resistance components between PF CFD and URANS CFD-based optimal designs for case 1

	Fr = 0.541				
	C _F	$C_{\rm PP}$	$C_{\rm PH}$	$C_{\rm P}$ ^a	Ст
Original model	2.918×10^{-3}	3.150×10^{-3}	-3.185×10^{-4}	2.831×10^{-3}	5.749×10^{-3}
Optimal model case 1 (PF CFD optimization)	2.976×10^{-3}	2.297×10^{-3}	4.793×10^{-4}	2.776×10^{-3}	5.798×10^{-3}
Optimal model case 1 (URANS CFD optimization)	2.981×10^{-3}	2.266×10^{-3}	-2.845×10^{-4}	1.981×10^{-3}	4.962×10^{-3}
	$\Delta C_{ m F}^{\ b}$	$\Delta C_{ m PP}{}^{ m b}$	$\Delta C_{ m PH}{}^{ m b}$	$\Delta C_{ m P}{}^{ m b}$	$\Delta C_{\mathrm{T}}^{b}$
Optimal model case 1 (PF CFD optimization) (%)	2.0	-27.1	250.5	-2.0	0.9
Optimal model case 1 (URANS CFD optimization) (%)	2.2	-28.1	10.7	-30.0	-13.7
	R _T ^b	Sinkage ^b	Trim ^b	S ^b	Disp. ^b
Optimal model case 1 (PF CFD optimization) (%)	108.2	38.4	77.5	107.3	100.6
Optimal model case 1 (URANS CFD optimization) (%)	90.3	49.5	72.6	104.6	100.0

Both evaluated by URANS CFD

^a $C_{\rm P} = C_{\rm PP} + C_{\rm PH}$

^b % original

PF CFD SBD may yield a curved keelline, but that is not allowed for URANS CFD SBD. In addition, a larger number of design variables is used for PF CFD SBD, while, without viscous effects of flow, PF CFD SBD explored a larger variation of designs, including somewhat excessive redesigns.

For the case 3 optimal designs, comparison between the two SBDs is not straightforward, since all designs in each Pareto set are solutions for each SBD. A focus here can be on comparison of F_2 -minimum design in each Pareto set, which is shown in Fig. 5 as case 3 NO 203 (Fig. 5a) and case 3 ID-556 (Fig. 5b) for PF CFD SBD and URANS CFD SBD optimal designs, respectively. Regarding reduction of F_2 , i.e., the seakeeping merit function, common trends for both SBD results are seen; i.e., compared with the F_1 -minimum design, the volume distribution is moved slightly inward to yield a more symmetric demihull

with respect to the keelplane, which is a conflicting trend with that for minimization of F_1 . Since F_1 is already significantly reduced, the aforementioned geometrical features for F_1 -minimum design partly remain.

To confirm the above-discussed trends in F_2 -minimum designs, URANS CFD SBD results for the MO ID-151 and MO ID-556 designs were compared. MO ID-151 is a F_1 - F_2 compromise design from the URANS CFD SBD Pareto set for case 3 problem. As shown in Fig. 5b, d, the MO ID-151 design is in between the F_1 -minimum and F_2 -minimum designs. F_1 and F_2 for the MO ID-151 design are smaller and larger than those for the MO ID-556 design (Table 3), respectively, consistent with the trends in wave field as shown in Figs. 19 and 20; i.e., the aforementioned trends for reduction of R_T are more significant for the former. Furthermore, Fig. 21 shows comparison of surface streamlines, pressure, and iso-Lamb scalar surfaces ($\Phi = 10$ isosurfaces

427
are displayed). It is seen that the generation of longitudinal vortices due to three-dimensional flow separation occurring near the keel for the original design is significantly reduced for the presented optimal designs, which also correlates with the reduction of pressure resistance for those designs.

Further investigation of the URANS CFD SBD/MO ID-151 design was performed by using the PF CFD method, and the results from both PF CFD and URANS CFD were compared (as summarized in Table 6). Notably, the reduction of $R_{\rm T}$ together with reduction of sinkage and trim values was confirmed by both CFD methods, although PF CFD predicts a smaller gain in $R_{\rm T}$, which can be related to the aforementioned lack of nonlinear free-surface effects and hydrostatic pressure component.



Fig. 19 Results for case 3 (MO ID-151 and 556). Comparison of wave profiles between the original and optimal designs. For URANS CFD-based optimization



Finally, the above discussions lead to the conclusion for CFD-based optimization that PF CFD is able to provide some important trends which will be useful to guide design optimization, especially as far as wave field optimization is concerned; however, the trend must be corrected by using higher-fidelity CFD method, e.g., URANS CFD, to yield more successful optimal design. Since it was considered that reduction of wavemaking resistance is the most essential task for all cases investigated in the present study, the modification trends predicted by PF CFD SBD are directly applied to define the type A modification method for URANS CFD SBD with a smaller number of design variables; and finally URANS CFD SBD is shown to yield a new design with significantly reduced $R_{\rm T}$. On the other hand, URANS CFD SBD results could be used to guide PF CFD SBD as well, which has not been considered yet in the present work. A more complete interactive approach where the systematic interaction between PF CFD SBD and URANS CFD SBD is implemented will realize a variable fidelity/variable physics approach in the ship hydrodynamic research area, and development of such an approach is our future work, as described later.

7 V&V procedures for evaluating optimization trends, and selection and EFD validation of optimal design

7.1 V&V procedures for evaluating optimization trends

The authors have proposed a verification and validation (V&V) procedure for a single-objective optimization results (Campana et al., 2006). This procedure represents an extension of one proposed by [21] for single numerical simulations and piggybacks on a validation approach for trends [22] by adding verification considerations. With reference to the uncertainties, we now need a condition that states that, if the expected improvement $\Delta_S = S_P - S_O$,





MO ID-151 MO ID-556

Fig. 21 Results for case 3 (MO ID-151 and 556). Comparison of surface streamlines, pressure, and iso-Lamb scalar surfaces. For URANS CFDbased optimization

 Table 6
 Comparison of resistance components for URANS CFD-based optimal design, MO ID-151 (case 3 optimal design from URANS CFD SBD)

	$\Delta C_{ m F}~^{ m a}$	$\Delta C_{ m P}$ a	$\Delta C_{ m T}$ $^{ m a}$	$\Delta R_{\mathrm{T}}^{\mathrm{a}}$	Sinkage ^a	Trim ^a
URANS CFD evaluation						
Optimal case 3 (MO ID-151) (%)	2.2	-30.0	-13.7	-9.3	-50.5	-27.4
PF CFD evaluation						
Optimal case 3 (MO ID-151) (%)	0.0	-15.5	-8.7	-4.7	-50.1	-42.1

Evaluations done by both URANS CFD and PF CFD

^a % original

where S_P and S_O are simulation values for the parent (original) and optimal designs, respectively, is greater than the simulation numerical noise U_{Δ_s} , then the optimized design is numerically verified, as follows:

Original

$$|\Delta_{\rm S}| > U_{\Delta_{\rm S}} = (U_{\rm SNP}^2 + U_{\rm SNO}^2)^{1/2}$$
(5)

where $U_{\rm SNP}$ and $U_{\rm SNO}$ are the simulation uncertainties for the parent and optimal designs, respectively, which may be assumed equal if the same numerical schemes are used. In a similar manner, if the improvement measured in the experiment, $\Delta_{\rm D} = D_{\rm P} - D_{\rm O}$, where $D_{\rm P}$ and $D_{\rm O}$ are experimental values for the parent and optimal designs, respectively, is greater than the experimental noise $U_{\Delta_{\rm D}}$, then the optimized design is experimentally verified, i.e.,

$$|\Delta_{\rm D}| > (U_{\rm D_P}^2 + U_{\rm D_O}^2)^{1/2} = U_{\Delta_{\rm D}}$$
(6)

The definition of the comparison error E now has to be modified into an *optimizer* error, E_{Δ} , focused on the trend. Hence, E_{Δ} can be defined as the difference between the measured and the expected improvements, $\Delta_{\rm D}$ and $\Delta_{\rm S}$: $E_{\Delta} = \Delta_{\rm D} - \Delta_{\rm S}$. The corresponding uncertainty equation is

$$U_{\rm E_{\Delta}} = \left(U_{\rm \Delta_{\rm D}}^2 + U_{\rm \Delta_{\rm S}}^2\right)^{1/2} \tag{7}$$

The last step is to state that, if the difference between the measured and the expected improvements is less than the uncertainty $U_{\rm E_A}$, that is

$$|E_{\Delta}| < U_{\mathrm{E}_{\Delta}},\tag{8}$$

then we may finally say that the optimized solution is validated at the interval $U_{E_{\Delta}}$.

7.2 Selection of optimal design for EFD campaign

In the following, selection of the optimal design for the experimental campaign is discussed. The above-described procedure to numerically verify the optimal design is applied for this purpose, to examine whether the improvement of the design is significant enough. The focus here is on the URANS CFD SBD/MO ID-151 design, a F_1 - F_2 compromise optimal design for the case 3 problem, which indicates considerable improvement over the original design, as confirmed by the PF CFD method as well as

the URANS CFD method. For the numerical verification, both URANS CFD and PF CFD results were used. The numerical uncertainties of the URANS CFD results were estimated by using three grids, i.e., 467,955, 2,352,735, and 3,690,420 points. The aforementioned $\Delta_{\rm S}$ and $U_{\Delta_{\rm S}}$ are shown in Tables 7 and 8, for the URANS CFD and PF CFD results, respectively.

For F_1 , the expected improvement Δ_S estimated by URANS CFD and PF CFD was 9.3% and 4.7%, respectively, and the simulation numerical noise U_{Δ_S} was 3.5% and 0.7%, respectively. Δ_S is larger than U_{Δ_S} for both URANS CFD and PF CFD, leading to the conclusion that the optimal design is numerically verified by both simulations for F_1 . For F_2 , the same scheme is used for both SBDs, and the expected improvement Δ_S is 40.4%. U_{Δ_S} for F_2 is not available, but the gain is considered to be significant based on the comparative study presented by [15].

In addition, values for sinkage and trim are considered. The expected improvement $\Delta_{\rm S}$ for sinkage estimated by URANS CFD and PF CFD was 50.4% and 58.3%, respectively, and the numerical noise $U_{\Delta_{\rm S}}$ was 7.5% and 0.7%, respectively. Therefore, $\Delta_{\rm S}$ is larger than $U_{\Delta_{\rm S}}$ for both simulations, and the optimal design is numerically verified for sinkage. The same conclusion is reached for trim, i.e., the $\Delta_{\rm S}$ estimated by URANS CFD and PF CFD was 27.4 and 54.7%, respectively, and $U_{\Delta_{\rm S}}$ was 7.1% and

Table 7 V&V procedure for multiobjective optimization results

0.7%, respectively; hence, $\Delta_{\rm S}$ is larger than $U_{\Delta_{\rm S}}$ for both simulations, and the optimal design is numerically verified for trim as well.

In summary, gains for the URANS CFD SBD/MO ID-151 design are numerically verified by both URANS CFD and PF CFD for F_1 , sinkage, and trim, and that for F_2 is considered to be significant enough. Finally, this design was selected for EFD validation, and further investigation on the gain and success of the optimization was conducted as described below.

7.3 EFD campaign to validate optimal design

An EFD campaign was performed at INSEAN towing tank by using 4-m models for both the original and optimal designs. The model size is compatible with the main dimensions of the INSEAN basin no. 2 (250 m long, 9 m wide, and 4.5 m deep) and avoids blockage effects. All resistance and seakeeping tests were carried out in this basin, equipped with a carriage capable of maximum speed of 10 m/s with precision of forward velocity of about 0.1%. During the tests, a load cell with reading range of 200 N was used. The same load cell was applied for both models and during the whole measurement campaign. Towing force was registered during the experimental tests, together with the bow and stern sinkage. Uncertainty analysis was

URANS CF	D							
Obj.	Model		S	D	E (%)	$U_{\rm D}~(\%)$	$U_{ m SN}~(\%)$	U _V (%)
F_1	Origina	al	8.80 (Kgf)	9.91 (Kgf)	-12.2	1.0	2.5	2.7
	Optima	al	7.98 (Kgf)	8.64 (Kgf)	-6.5	1.0	2.5	2.7
F_2	Origina	al	0.99	2.41	-65.4	-	-	-
	Optima	al	0.59	2.14	-71.4	-	-	-
	$\Delta_{\rm S} = S$	$S_{\rm P} - S_{\rm O}$	$\Delta_{\rm D} = D_{\rm P} - {\rm D}_{\rm O}$	$U_{\Delta_{ m S}}$	$U_{\Delta_{ m D}}$	$ E_{\Delta} $	$U_{\mathrm{E}_{\Delta}}$	
F_1 (%)	9.3		12.8	3.5	1.4	3.5	3.8	
$F_2(\%)$	40.4		11.2	_	-	29.2	_	
URANS CF	D							
		Model	S	D	E (%)	$U_{\rm D}~(\%)$	$U_{\rm SN}$ (%)	U _V (%)
Sinkage (%	$%L_{\rm PP}$)	Original	-0.00248	-0.0048	-12.2	1.0	5.3	5.4
		Optimal	-0.00123	-0.0021	-6.5	1.0	5.3	5.4
Trim (°)		Original	1.824	1.72	6.0	1.0	5.0	5.1
		Optimal	1.324	1.04	27.3	1.0	5.0	5.1
		$\Delta_{\rm S} = S_{\rm P} - S_{\rm P}$	$S_{\rm O}$ $\Delta_{\rm D}$ =	$= D_{\rm P} - D_{\rm O}$	$U_{\Delta_{ m S}}$	$U_{\Delta_{ m D}}$	$ E_{\Delta} $	U_{E_Δ}
Sinkage (%))	50.4	56.3		7.5	1.4	5.8	7.6
Trim (%)		27.4	39.5		7.1	1.4	12.1	7.2

For URANS CFD results for the original design and case 3 optimal design (MO ID-151)

PF CFD								
Obj. 1	Model	S		D	E (%)	$U_{\rm D}~(\%)$	$U_{\rm SN}~(\%)$	U _V (%)
F_1	Original	10.6 (K	(gf)	9.91 (Kgf)	7.0	1.0	0.5	1.1
(Optimal	10.1 (K	(gf)	8.64 (Kgf)	14.7	1.0	0.5	1.1
F_2 (Original	0.99		2.17	-54.4	-	-	-
(Optimal	0.59		1.99	-70.4	-	-	-
Obj.	$\Delta_{\rm S} =$	$= S_{\rm P} - S_{\rm O}$	$\Delta_{\rm D}$ =	$= D_{\rm P} - D_{\rm O}$	$U_{\Delta_{ m S}}$	$U_{\Delta_{ m D}}$	$ E_{\Delta} $	$U_{\mathrm{E}_{\Delta}}$
F_1 (%)	4.7		12.8		0.7	1.4	8.1	1.6
F_2 (%)	40.4		11.2		-	-	29.2	_
PF CFD								
	1	Model	S	D	E (%)	$U_{\rm D}~(\%)$	$U_{ m SN}~(\%)$	U_{V} (%)
Sinkage (%L _{PP})	(Original	-0.0056	-0.0048	-12.2	1.0	0.5	1.1
	(Optimal	-0.0028	-0.0021	-6.5	1.0	0.5	1.1
Trim (°)	(Original	2.24	1.72	30.2	1.0	0.5	1.1
	(Optimal	1.30	1.04	25.0	1.0	0.5	1.1
		$\Delta_{\rm S} = S_{\rm P} - S_{\rm O}$	Δ	$A_{\rm D} = D_{\rm P} - D_{\rm O}$	$U_{\Delta_{ m S}}$	$U_{\Delta_{ m D}}$	$ E_{\Delta} $	$U_{\mathrm{E}_{\Delta}}$
Sinkage (%)		58.3	5	6.3	0.7	1.4	2.1	1.6
Trim (%)		54.7	3	9.5	0.7	1.4	15.1	1.6

Table 8 V&V procedure for multiobjective optimization results

For PF CFD results for the original design and case 3 optimal design (MO ID-151)

performed for total resistance only for the design speed of Fr = 0.541, recording ten different resistance time histories.

The success of the optimization processes was confirmed by the experimental measurements. Figure 22 shows EFD results for comparison of total resistance $R_{\rm T}$, sinkage, and trim between the original and optimal designs. At the design speed of Fr = 0.541, those values were successfully reduced for the optimal design, i.e., gains of $R_{\rm T}$ (i.e., F_1 for case 3), sinkage, and trim are 12.8%D (% data for the original design), 56.3%D, and 39.5%D, respectively. It is also shown that the second objective function F_2 is significantly reduced, i.e., 11.2%D. As a positive side-effect, the reduction of $R_{\rm T}$ occurred in an extended range of Fr around the design speed, i.e., 0.4 < Fr < 0.7, where Fr = 0.7 is the maximum Fr used in the EFD; for example, the reduction of $R_{\rm T}$ at Fr = 0.460and Fr = 0.622 (i.e., lower and higher Fr for case 2 problem) was 9.9%D and 10.5%D, respectively.

7.4 Validation of optimal results

Finally, the V&V analysis is summarized in the following. As already discussed, simulations from both URANS CFD and PF CFD were considered, and the results are shown in Tables 7 and 8, respectively. Some of the cells in the tables are empty because uncertainties in the data are currently not available for F_2 due to the nature of the seakeeping test itself.

 $U_{\rm D}$ for F_1 was estimated to be 1.0%D for both the original and optimal designs. The actual improvement $\Delta_{\rm D}$ (12.8%D) is larger than $U_{\Delta \rm D}$ (1.4%D), hence the optimal design is experimentally verified for F_1 . F_2 also indicates significant reduction, i.e., $\Delta_{\rm D}$ is 11.2%D. The $\Delta_{\rm D}$ for sinkage and trim is 56.3%D and 39.5%D, respectively, while $U_{\Delta \rm D}$ for those values is estimated to be 1.4%D and 1.4%D, respectively; hence, the optimal design is experimentally verified for sinkage and trim. For both SBDs, the comparison error E for F_1 , sinkage, and trim was larger than the validation uncertainty $U_{\rm V}$. Moreover, the trend of optimization for F_1 for URANS CFD is validated; i.e., E_{Δ} is 3.5%D and $U_{\rm E\Delta}$ is 3.8%D, i.e., E_{Δ} is smaller than $U_{\rm E\Delta}$ so the optimal result is validated for F_1 at the level of $U_{\rm E\Delta}$.

In summary, trends for total resistance, sinkage, and trim between the original and URANS CFD SBD/MO ID-151 optimal design were numerically and experimentally verified by the present EFD and CFD methods. The trends in the optimal results were correctly predicted by both CFD methods, and the URANS CFD SBD design was validated in terms of total resistance at the level of $U_{\rm E\Delta}$. To complete the validation of each variable, higher-resolution computational grids will be needed to resolve flow complexities, particularly for the present high-speed catamaran applications.



Fig. 22 EFD results. Comparison of total resistance, sinkage, and trim between the original and optimal designs

8 Concluding remarks

Numerical optimization of the initial design of a fast catamaran, i.e., HSSL-B, was carried out through SBD frameworks, based on advanced free-surface URANS and PF solvers and GO algorithms. Three optimization problems, including single- and multiobjective optimization problems, proposed by designers from Bath Iron Works, were successfully solved, and finally an experimental campaign was carried out to validate the optimal design. Those results were evaluated by using a new V&V methodology for assessing uncertainties and errors in simulation-based optimization.

Based on numerical verification to identify meaningful improvements over the original design, the URANS CFD SBD/MO ID-151 optimal design for case 3 problem was selected for an EFD campaign. The success of the optimization processes was confirmed by the EFD results. At the design speed of Fr = 0.541, values for the optimal design were successfully reduced; and through the aforementioned V&V procedure, the trends in total resistance, sinkage, and trim between the original and optimal designs were numerically and experimentally verified, and the trend in total resistance was validated.

In the present study, PF CFD SBD and URANS CFD SBD were used in a complementary manner. Since PF CFD SBD is less expensive, a greater number of design variables was used to explore even somewhat drastic redesigns. Then, the results were used to guide the more expensive URANS CFD SBD, which uses fewer design variables. Through practicing this approach, we reached the conclusion for CFD-based optimization that PF CFD is able to provide some important trends which will be useful to guide design optimization, especially as far as wave field optimization is concerned; however, the trend must be corrected by using higher-fidelity CFD methods, e.g., URANS CFD, to yield more successful optimal design. An SBD where the systematic interaction between PF CFD SBD and URANS CFD SBD is implemented will realize a variable fidelity/variable physics approach in the ship hydrodynamic research area. The idea of combining lowfidelity models together with occasional (heuristic) recourse to high-fidelity models (for monitoring the progress of the algorithm) is not new (e.g., [23]). Here, we want to introduce a new algorithm for optimizing a nonlinear function whose first-order derivatives do not exist, or exist but are unavailable.

Hence, in our future work, further development of our SBD is also of great interest, such as introduction of the above-mentioned physics-based variable-fidelity RANS/ PF approach, implementation of a complete interface among the two optimization methods developed in the present work, free-surface URANS solvers, computer-aided design (CAD) systems, and demonstrations for waterjet/hull optimization problems. Finally, the ongoing transition of our SBD to the ship building industry and research institutions must be completed (for current status, see [1]).

Acknowledgments This work has been partially supported by the U.S. Office of Naval Research under grants N00140210256, N000140510617, N000140210304, and N000140510723, through Dr. Pat Purtell. The authors would like to express their appreciation for his kind support and encouragement. The authors' appreciation is extended to Dr. C. Carey and Mr. R. Hoffman (Bath Iron Works) for HSSL-B geometry and optimization test problems.

References

- Stern F, Carrica P, Kandasamy M, Ooi SK, Gorski J, O'Dea J, Fu T, Hendrix D, Kennell C, Hughes M, Miller R, Marino T, Hess D, Kring D, Milewski W, Hoffman R, Smith S, Cary C, Taylor P, Peterson B, Harris D, Monaco C (2008) Computational hydrodynamic tools for high-speed sealift: phase II final report. IIHR Technical Report, No. 465, IIHR-Hydroscience & Engineering, College of Engineering, The University of Iowa, Iowa City, IA, USA
- Stern F, Carrica P, Kandasamy M, Gorski J, O'Dea J, Hughes M, Miller R, Hendrix D, Kring D, Milewski W, Hoffman R, Cary C (2007) Computational hydrodynamic tools for high-speed transports. Trans SNAME 114:55–81
- 3. Doctors LJ, Scrace RJ (2003) The optimization of trimaran sidehull position for minimum resistance. FAST 2003, Ischia, Italy
- Kennell C (2004) Model test result for a 55 knot high-speed sealift trimaran. RINA design and operation of trimaran ship conference, London, pp 28–29
- Campana EF, Peri D, Pinto A, Tahara Y, Stern F (2006) Shape optimization in ship hydrodynamics using computational fluid dynamics. Comput Methods Appl Mech Eng 196:634–651
- Tahara Y, Peri D, Campana EF, Stern F (2008) Computational fluid dynamics-based multiobjective optimization of a surface combatant. J Mar Sci Technol 13(2):95–116
- Peri D, Campana EF (2005) High-fidelity models and multiobjective global optimization algorithms in simulation based design. J Ship Res 49(3):159–175
- Molland AF, Wellicome JF, Couser PR (1996) Resistance experiments on a systematic series of high speed displacement catamaran forms: variation of length-displacement ratio and breadth-draught ratio. RINA Trans 138:55–68
- 9. Armstrong T (2003) The effect of demihull separation on the frictional resistance of catamarans. FAST 2003, Ischia, Italy

- Campana EF, Fasano G, Pinto A (2006b) Globally convergent modifications of particle swarm optimization for unconstrained optimization. INSEAN Tech Report 2006-20
- Campana EF, Fasano G, Pinto A (2006c) Dynamic system analysis for the selection of parameters and initial population in particle swarm optimization. INSEAN Tech Rep. 2006-56
- Peri D, Pinto A, Campana EF (2006) PSO-DRAGO hybrid algorithm for global optimization. INSEAN technical report 2006-054
- Bassanini P, Bulgarelli UP, Campana EF, Lalli F (1994) The wave resistance problem in a boundary integral formulation. Surv Math Ind 4:151–194
- 14. Peri D, Roccaldo R, Franchi SI (1998) Influence of the submergence and the spacing of the demihulls on the behaviour of multihulls marine vehicles: a numerical application. PRADS '98, Den Haag, The Netherlands
- Lugni C, Colagrossi A, Landrini M, Faltinsen OM (2004) Experimental and numerical study of semi-displacement mono-hull and catamaran in calm water and incident waves. 25th Symposium on naval hydrodynamics, 8–13 August, St. John's, Canada
- Tahara Y, Stern F, Himeno Y (2004) Computational fluid dynamics-based optimization of a surface combatant. J Ship Res 48(4):273–287
- Tahara Y, Tohyama S, Katsui T (2006) CFD-based multiobjective optimization method for ship design. Int J Numer Methods Fluids 52:449–527
- Ono I, Kobayashi S (1997) A real-coded genetic algorithm for function optimization using unimodal normal distribution crossover. 7th International Conference on genetic algorithms, pp 246–253
- 19. Deb K (2001) Multi-objective optimization using evolutionary algorithms. Wiley, New York
- Carrica P, Wilson RV, Stern F (2006) Unsteady RANS simulation of the ship forward speed diffraction problem. Comput Fluids 35:545–570
- Stern F, Wilson RV, Coleman HW, Paterson EG (2001) Comprehensive approach to verification and validation of CFD simulations—part 1: methodology and procedures. J Fluids Eng 123:793–802
- Coleman HW, Stern F (1997) Uncertainties in CFD code validation. ASME J Fluids Eng 119:795–803
- Alexandrov NM (1996) A trust region framework for managing approximation models in engineering optimization. 6th AIAA/ NASA/ISSMO Symposium on multidisciplinary analysis and opt., AIAA-96-4102

International Shipbuilding Progress 60 (2013) 277–308 DOI 10.3233/ISP-130098 IOS Press

Simulation based design optimization of waterjet propelled Delft catamaran

Manivannan Kandasamy^a, Daniele Peri^b, Yusuke Tahara^c, Wesley Wilson^d, Massimo Miozzi^b, Svetlozar Georgiev^e, Evgeni Milanov^e, Emilio F. Campana^b and Frederick Stern^{a,*} ^a *IIHR*, *Hydroscience and Engineering, University of Iowa, Iowa City, USA*

^b INSEAN-CNR, National Research Council, Maritime Research Centre, Rome, Italy

^d NSWCCD, Naval Surface Warfare Center Carderock Division, Bethesda, MD, USA

^e BSHC, Bulgarian Ship Hydrodynamics Center, Varna, Bulgaria

The present work focuses on the application of simulation-based design for the resistance optimization of waterjet propelled Delft catamaran, using integrated computational and experimental fluid dynamics. A variable physics/variable fidelity approach was implemented wherein the objective function was evaluated using both low fidelity potential flow solvers with a simplified *CFD* waterjet model and high fidelity *RANS* solvers with discretized duct flow calculations. Both solvers were verified and validated with data for the original hull. The particle swarm optimizer was used for single speed optimization at Fr = 0.5, and genetic algorithms were used for multi speed optimization at Fr = 0.3, 0.5 and 0.7. The variable physics/variable fidelity approach was compared with high fidelity approach for the bare-hull shape optimization and it showed an overall *CPU* time reduction of 54% and converged to the same optimal design at Fr = 0.5. The multi-speed optimization showed design improvement at Fr = 0.5 and 0.7, but not at Fr = 0.3 since the design variables were obtained based on sensitivity analysis at Fr = 0.5. High fidelity simulation results for the optimized barehull geometry indicated 4% reduction in resistance and the optimized waterjet equipped geometry indicated 11% reduction in effective pump power required at self-propulsion. Verification was performed for the optimized hull form and its reduction in powering will be validated in forthcoming experimental campaign.

Keywords: Simulation-based design, ship design, waterjet propulsion, variable fidelity

1. Introduction

Waterjet (WJ) propulsion features shallow draft design, smooth engine load, less vibration, lower water borne noise, no appendage drag, better efficiency at high speeds and good maneuverability. Pre-designed waterjets are readily available for any type of vessel based on the engine power, resistance curves, and the design speed of the ship. However, the performance of the WJ systems with respect to inlet efficiency, velocity distribution at the impeller plane, and cavitation inception at

^c NMRI, National Maritime Research Institute, Tokyo, Japan

^{*}Corresponding author. E-mail: Frederick-stern@uiowa.edu.

⁰⁰²⁰⁻⁸⁶⁸X/13/\$27.50 © 2013 - IOS Press and the authors. All rights reserved

cutwater, rely on the inlet velocity ratio (*IVR*), which depends on the specific hull shape in addition to the speed of operation. Optimization of the pre-designed waterjets with regard to the specific hull forms would considerably improve the powering performance.

Recent developments in *CFD* and high performance computing have significantly advanced simulation-based design (*SBD*) optimization of ship hull forms and greatly reduced the cost incurred by traditional build and test approaches. Campana et al. [5] provides an overview of all the relevant aspects involved in the development of an up-to-date *SBD* framework for ship design: optimization algorithms to explore the design space in search for the optimum design, automatic mesh and geometry modifiers and analysis tools for evaluating the objective function and drive the optimization algorithm.

The analysis tools used for SBD optimization vary in levels of approximation, progressing from linear 2D slender body theory, to non-linear 3D panel methods, to detailed RANS calculations including all ship appendages. Stern et al. [26] gives a detailed overview of the different fidelity codes used at different stages of the SBD process. Combinations of different methods, namely "zonal" or "domain decomposition" approaches have also been developed. Janson and Larson [14] divided the domain into three zones for the simulation-based optimization of a series 60 hull. The first zone covers the entire hull and a part of its surrounding free-surface and uses a free-surface Rankine-source type potential-flow method. The second zone is a thin layer at the hull surface and uses a momentum integral type boundary layer method. The third zone includes the aft part of the hull and uses RANS. The zones are computed in sequence and boundary conditions are generated for succeeding zones. This approach reduces the computational time considerable compared to a full RANS solver. However, since the viscous non-linear effects on the waves were neglected, experiments carried out for the original and the optimized hull did not validate the resistance improvements obtained in the computations. The computations predicted the right trend, but underpredicted the wave resistance magnitude. Consequently, the authors recommend the use of a Navier-Stokes solver with free-surface calculations for future work. Similar observation was made by Kandasamy et al. [17] where the early stage potential flow optimization predicted the correct trend but underpredicted the wave resistance magnitude for a foil-assisted semi-planning catamaran.

Numerical optimization algorithms for automatic optimal design are independent of the flow-solver complexity, which is however an issue (and has to be properly treated via variable fidelity and/or metamodels approaches, see Campana et al. [5]) when the computational cost is large. Gradient based optimization methods are classified as local optimization schemes since they require the calculation of the gradient of the objective function and hence may get stuck in a local minimum. Meta-heuristic optimization methods such as genetic algorithms and particle swarm optimizers are classified as global optimization algorithms since they are derivative-free and less prone to getting stuck in a local minimum. However, they are much more expensive in terms of number of objective function evaluations needed for the achievement of

the convergence of the algorithm, and they are typically very slow to converge at the precise optimal value. Finally, the way in which the automatic geometrical and volume (or surface) grid changes are handled is the third important block of any *SBD* tool. The computational grid adopted in the analysis must be deformed, in background, each time there is a new perturbed design to be evaluated, trying to preserve at the same time the quality of the new mesh. In conjunction with *RANS* solvers, regridding issues may become extremely relevant to the performance and the final result of the optimization.

The main objective of the current hydrodynamic optimization was to implement a multi-pronged parallel optimization for powering optimization of a WJ-propelled catamaran at Fr = 0.5, using different features of the SBD toolbox. A complementary multi-speed optimization study was also carried out for three different speeds, Fr = 0.3, 0.5 and 0.7. The toolbox used for the optimization is a product of the longterm ongoing collaboration between IIHR, INSEAN and NMRI research groups. The toolbox consists of the high fidelity (HF) URANS solver CFDShip [6], and the low fidelity (LF) linearized potential flow solver WARP [2], two evolutionary optimization algorithms, namely a multi-objective genetic algorithms - MOGA [29] and a particle swarm optimization (PSO) method [5]. It also contains different geometry modification tools and meta-modeling techniques, namely a free form deformation (FFD) approach [4] and different morphing techniques. Previous versions of the toolbox have been successfully used for progressively complex designs, namely, mono-hull surface combatant [4], multi-hull high speed sea lifts [29], SWATH displacement ships [28], foil-assisted semi-planing catamaran ferries [17] and uncoupled barehull and WJ inlet optimization of JHSS mono-hull [18]. The current study extends the SBD toolbox to the water-jet propelled Delft catamaran (DC) that has pronounced WJ-hull interaction caused by the hull contour at inlet, unlike JHSS that has a flat intake. The absence of gooseneck and multiple adjacent waterjets makes it more costeffective for forthcoming build and test validation of the optimized hull form. The multi-hull geometry provides additional opportunities to explore asymmetric effects of the demi-hulls on wave interference and inlet air-entrainment. Sufficient model testing data of the original hull is available for code validation and data has already been used as a benchmark for many hydrodynamic performance and validation studies; e.g., effects of separation distances [12], sea-keeping [7], manoeuvring [19], and also for the development of the integral force/moment CFD WJ model [16].

Since viscous effects involving boundary layer ingestion play a significant part in the *WJ*-hull interaction efficiency and the *WJ* inlet ducting efficiency, potential flow methods cannot be used for the duct inlet optimization. Hence, a variable physics approach is used for initial hull-form optimization with the simplified *CFD WJ* model to replicate the effects of the *WJ* without simulating the duct flow. The best performing hull is then attached with the discretized *WJ* duct for detailed inlet optimization using the high fidelity *RANS* solver with an actuator disk model to replicate the effects of the impeller.

2. methodology

The *SBD* methodology comprises of three main parts: the optimizer, the geometry modification methods and the analysis tools (in the case of fluid dynamic optimization the latter are flow solvers, Fig. 1). The analysis tools send the evaluated objective functions for a certain set of design variables to an optimizer, which searches for their minimum value under the general non-linear programming mathematical framework and continually updates the design variables. Geometry modeling methods provides the link between the two by deforming the hull shape based on the updated design variables.

2.1. Optimizers

The *SBD* toolkit features two derivative-free, global optimization algorithms: PSO is a single/multi-objective global optimization algorithm and MOGA that is primarily a multi-objective optimization algorithm, but can also be used for single objective problems by setting the second objective function equal to the first one. MOGA is a stochastic algorithm, since the final results depends on the random variables involved in the initial formulation, whereas a new deterministic version of the original stochastic.

2.2. Geometry modification methods

Three different options are available for parametric design: B-splines, FFD technique and morphing. With B-splines, the hull form deformation is controlled by the position, direction and magnitude of the knot vectors that form the basis of the design variables. With FFD, the hull form is embedded into a parallelepiped, split by a number of coordinate planes. Crossings between the planes are the control points of the FFD, and the movement of the control-points results in the deformation of the volume, and the embedded hull. With morphing [24], the grid points are obtained as the weighted sum of the corresponding corners of the base grids, and weights are represented by the design variables of the optimization problem.



Fig. 1. SBD toolbox and methodology.

2.3. Solver methods

Two options are available for the choice of flow solver CFDShip-Iowa developed at IIHR is used as HF analysis tool, and the potential flow solver WARP, developed at INSEAN is adopted as LF code.

2.3.1. CFDShip-IOWA

The URANS solver uses a single-phase level set method to predict the free surface. A second order upwind scheme is used to discretize the convective terms of momentum equations and a pressure-implicit split-operator algorithm is used to enforce mass conservation on the collocated grids. The pressure Poisson equation is solved using the *PETSc* toolkit. All the other systems are solved using an alternating direction implicit method. For a high performance parallel computing, a MPI-based domain decomposition approach is used, where each decomposed block is mapped to one processor. A simplified body force model is used for *WJ* simulation to prescribe axisymmetric body force within the duct.

2.3.2. WARP

The WARP potential code is a classical boundary-element method solver. To solve numerically the set of integral equations arising from the Laplace equation arising from the potential flow description of the problem, the wetted hull surface and a region of the free surface are discretized into plane elements of quadrilateral shape. The free surface boundary conditions are linearized around the calm water level. Derivatives of the velocity potential are obtained analytically. The viscous terms are not directly considered during the solution, and an accurate estimate of the wave resistance is obtained by means of a locally adapted *ITTC* formula, where a local Reynolds number is used for each panel, based on the local velocity. The code allows for free sinkage and trim: the hull position is obtained by the simplified *CFD WJ* model.

2.3.3. Water-jet models

The *WJ* duct flow can either be simulated through a discretized duct with an embedded actuator disk model to replace the pump system, or by using the simplified *CFD WJ* model to replace the whole duct control volume.

Simulations with embedded actuator disk model for WJ propelled JHSS have been validated recently by Delaney et al. [9] without free-surface calculations, and Takai et al. [31] who included free-surface calculations. Delaney et al. [9] indicated that the exclusion of the shaft in the actuator model had negligible influence on the performance analysis of the WJ system and they showed less than 1% error for self-propulsion flow rate, whereas Takai et al. [31] showed a 6% error which is attributed to overset grid interpolation errors within the duct.

The simplified CFD WJ model is used here to incorporate the effects of the WJ induced vertical forces and pitching moments on the sinkage and trim of the hull,

M. Kandasamy et al. / Simulation based design optimization



Fig. 2. CFD Waterjet model control volume.

without requiring detailed simulations for the *WJ* duct flow during the bare-hull optimization. The model uses the control volume shown in Fig. 2, which is selected with consideration to implementation simplicity in *CFD* using the same bare-hull grid by representing the *WJ* system by vertical reaction forces and pitching reaction moment, and by representing the *WJ*/hull interaction using a vertical stern force. The detailed duct flow calculations from the original hull provide the *WJ* induced vertical forces and pitching moments to be used during the hull shape optimization. Details of the model are provided in Kandasamy et al. [16] who validated its usage for *WJ* propelled DTMB-5594. Detailed waterjet flow simulation results from JHSS and *DC* were also used to investigate feasibility of deriving correlations based on the *WJ* geometry and *IVR* at working point. Ultimately, the model will be of most use if these generalized correlations can be obtained, which would provide the *WJ* induced forces and moments without prior experiments and/or detailed simulations of the duct flow.

2.4. Variable fidelity/variable physics approach

Numerical optimization in the naval field is shifting toward the use of High Fidelity (HF) *CFD* solvers, increasing the level of the physical content of the applied mathematical models adopted to guide the optimizer. This is done at the expense of computational power required for the analysis of the different designs, required during the course of the optimization.

One possible strategy to reduce the computational effort is to reduce the calls to the HF solver using approximation and/or interpolation models. A limited number of expensive HF simulations that sample the design space by some proper technique is used to obtain an approximation of the objective function (a meta-model, e.g. [23]). The optimization is then carried out using the meta-model whereas the optimum is checked using the HF code. The accuracy of the meta-model is then increased with every new point and the process is iterated until some kind of convergence is obtained.

The main criticism formulated for the use of meta-models is based on the difficulties in deriving a reliable approximation of the objective function when the number of variables is not small. An attractive alternative is based on the idea of exploiting two

or more physical models of different complexity (fidelity) for computing the same objective function. A Low Fidelity (LF) solver, fast but relatively simple, is adopted in conjunction with a High Fidelity (HF) solver, more reliable but time consuming. This simple – and old idea in engineering approaches to solve complex problems – is then "reinforced" with a solid mathematical background for the determination of the *correlation law* between HF and LF that gives an answer to the obvious question of *when* to switch from HF to LF and vice-versa. This represent the core of the *VF/VP* approach. The problem is solved using the fast LF tool, and the *trust region* approach (the mathematical theory) gives the rules for the systematic switch from LF to HF. Large *CPU* time savings (of the order of 50% and more, depending of the application) are obtained while the consistency between the two formulations (with and without *VF/VP*) is guaranteed.

The first proposal of the formulation of the VF/VP framework was presented for gradient based approaches and local optimization problems [1]. The essence of the idea stems from the basic Taylor series: any continuous and differentiable function of N variables can be locally expanded in Taylor series, so that in the neighborhood of the computational point x^0 holds the relationship:

$$f(x) = f\left(x^{0}\right) + \sum_{i=1}^{N} \frac{\partial f}{\partial x_{i}} \left(x_{i} - x_{i}^{0}\right) + \mathcal{O}^{2}.$$
(1)

Now, if we have two different models (LF and HF) to compute the same objective function, we can define a gap function as the difference of the two values at any x point of the *N*-design space:

$$\beta(x) = f^{\text{HF}}(x) - f^{\text{LF}}(x).$$
(2)

So that:

$$f^{\rm HF}(x) = \beta(x) + f^{\rm LF}(x). \tag{3}$$

From Eq. (3) follows that *if* the value of $\beta(x)$ is known over all the space, the HF would have been not needed anymore. The value of the *LF* function plus the gap function β would have then given the correct HF value. The problem is therefore shifted on the computation/evaluation of $\beta(x)$, in which the Taylor series intervenes. If we apply a Taylor expansion of β around the current design point x^0 , and we stop at the first order as in previous equation, we obtain an approximation of β , say β^T . Obviously, β^T is exact only at the linearization point x^0 , while some discrepancies arise once we move away from x^0 . We call *trust region* the portion of space in which the β^T model is considered *trustable*, i.e. sufficiently accurate. This region is assumed spherical and centered in the linearization point x^0 and with *trust-region* radius ρ that can be adjusted dynamically.

284

The missing quantities to compute β^T are the value of $\beta(x^0)$ and the derivatives of $\beta(x)$ with respect to the design variables. If we are using a derivative-based optimization algorithm, these quantities are needed in any case, and we can compute them at the first step of the optimization algorithm. Since then, we are using the LF values plus correction, and we check the correctness of the approximation β^T by comparing the true HF value and the approximated $HF^T = \beta^T + LF$. In a derivative-based algorithm, two different phases are identified: firstly, the gradient of the function is computed, and it is applied as is, or combined with previous values, in order to detect a descent direction. After that, a line search is performed along the descent direction and the minimum value of the objective function is identified along this line. All the computations required in these phases are computed by using LF prediction plus β^T . At the end of the line search, HF is computed and compared with HF^T, and the relative difference is evaluated as

$$r = \frac{\mathrm{HF}_2 - \mathrm{HF}_1}{\mathrm{HF}_2^T - \mathrm{HF}_1^T}.$$
(4)

Where subscripts 1 and 2 indicate respectively the initial and final point of the line search: r represents the ratio between the real and estimated improvement. We can call it consistency check, since this is checking the consistency between the full HF problem and the *VF/VP* problem. If r is sufficiently close to the unit value, we can trust the local model β^T inside this region and the trust region radius can be increased. On the contrary, this is a sign that we cannot trust the model up to that distance, and the radius of the region is to be reduced: the step length of the line search is reduced, and another solution is computed, and the consistency check repeated. If the check fails repeatedly, and becomes too small, the model is recomputed and re-initialized.

This classical formulation is well suitable for local optimization methods based on derivatives. Results for ship design applications are reported in [23], where savings of about 75% are obtained. However, if a global optimization algorithm is applied, and derivatives are not computed, the framework is no longer convenient, since the initialization of the approximated model β^T requires the derivatives of both the LF and HF objective function. Consequently, a global approximation/interpolation model of the scaling factor β is needed: in this case, kriging interpolation model is applied. The trust region radius is no longer a singled value function, but depends on the computational point. Since we are using PSO algorithm for optimization, we can assign a different ρ^i for each (*i*th) particle of the swarm. Each time both HF and LF are computed, the consistency check is performed. If the new position of the particle is located inside the trust region, HF is not computed, and only LF plus correction would replace one of the attractors of the swarm, and we must verify the accuracy of the prediction by $\beta^T + LF$.

-472-

3. Experimental methods

3.1. Model design and test plan

BSHC constructed a *DC* model with the main particulars tabulated in Table 1 and equipped it with available stock *WJ*.

Initial testing showed excessive swirl and immersion of the nozzle, which was rectified by adding extra fins at the stator and reducing the nozzle diameter. The new design showed significant reduction in swirl and nozzle immersion. The *DC* model was then shipped to INSEAN, who repeated the self-propulsion tests as a precursor to forthcoming validation tests on the optimized design.

For towed bare hull (BH) experiments, the difference between the two facilities was the location of the vertical pivot point. BSHC had a pivot point located above the LCG = 1.91 m with KG = 0.28 m, whereas INSEAN used hinges at FP and AP and adjustable height sliding towing post which allowed the pitch motions to be centered about the center of gravity during self-propulsion tests, i.e., LCG = 1.91 m, KG = 0.34 m.

For self-propelled (SP) experiments, BSHC assumed symmetry and performed measurements only on the starboard side *WJ*, whereas INSEAN performed measurements on both sides to account for installation uncertainty between the port and the starboard waterjets. Both facilities followed the ITTC [13] procedural guidelines described in detail in [20] and [21].

3.2. Overview of WJ test procedures

The ITTC Propulsion Committee [13] recommends the 'momentum flux method' using control volume analysis for prediction of the powering performance of a WJ

Table 1

Particulars of the DC geometry					
Main particulars	Symbol	Model			
Length overall, m	L_{OA}	3.8220			
Length between perpendiculars, m	$L_{\rm PP}$	3.6274			
Length on waterline, m	$L_{\rm WL}$	3.6274			
Breadth moulded, single hull, m	B	0.2904			
Clearance b/n hull CPs, m	-	0.8470			
Draft at FP, m	T_{F}	0.1815			
Draft at AP, m	$T_{\rm A}$	0.1815			
Displacement volume, m ³	Δ	0.0770			
Prismatic coefficient*	$C_{\rm P}$	0.6160			
Block coefficient*	$C_{\mathbf{B}}$	0.4027			
Longitudinal C.B.**	LCB	-0.0970			
Wetted surface area (bare hull), m ²	5	1.4220			



M. Kandasamy et al. / Simulation based design optimization

286

Fig. 3. Control volume for momentum-flux method: (a) EFD stations, (b) CFD inflow boundary shape. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)

driven ship [32]. The control volume shown in Fig. 3(a) is defined by a streamtube consisting of the inlet (AB), an upstream imaginary surface in the flow through which it is assumed no mass transport occurs (BC), ducting system, pump, nozzle and outlet. The control volume boundaries capture all inflow, outflow of WJ system, and provide ease of measurement of volume flow-rate (Q_{SP}), and momentum and energy fluxes.

For all tested craft speeds, the inflow boundary was assumed rectangular and its size was determined from BH inlet velocity-field measurements. Fig. 3(b) shows the *CFD* inflow boundary at Fr = 0.5, which is elliptical similar to previous simulations [31]. Van Terwisga [32] concluded that the inlet capture area for Athena was also elliptical, but the shape does not have significant effect on the ingested momentum and energy flux. The net jet thrust (T_{NET}) of the *WJ* system is obtained from the net rate of change of momentum over the control volume.

The WJ-hull system is decomposed into a BH system and a WJ system. This facilitates independent evaluation of the pump efficiency (η_{pump}) , ducting efficiency (η_{duct}) , jet efficiency (η_{jet}) , and the thrust deduction factor (1 - t) that constitute the overall WJ system efficiency (η_d) . Note that the last term (1 - t) can be alternatively expressed as a product of wake fraction (1 - w), and hull efficiency $\eta_H = (1 - t)/(1 - w)$, but were not essential for the present application, which focusses mainly on η_{duct} .

$$\eta_d = \eta_{\text{pump}} \times \eta_{\text{duct}} \times \eta_{\text{jet}} \times (1 - t).$$
(5)



Fig. 4. Waterjet system decomposition.

Figure 4 illustrates the flow of energy from the prime mover to the BH system through the WJ system, which comprises of the pump, duct, and jet systems. The measured BH resistance (R_{BH}) and T_{NET} at SP velocity (V_{Ship}) allow for the calculation of P_E and P_{TE} , respectively.

$$P_{\rm E} = R_{\rm BH} \times V_{\rm Ship},\tag{6}$$

$$P_{\rm TE} = T_{\rm NET} \times V_{\rm Ship}.\tag{7}$$

 P_{JSE} is obtained from energy flux difference between station 7 (*E*7) and station 1 (*E*1), and P_{PE} is obtained from energy flux difference between station 5 (*E*5) and station 3 (*E*3).

$$P_{\rm JSE} = E7 - E1,\tag{8}$$

$$P_{\rm PE} = E5 - E3. \tag{9}$$

Table 2							
Powering and motions $U_{\rm FB}$							
		BH			SP		
	$R_{\rm BH}$	$\sigma_{\rm BH}$	$ au_{ m BH}$	$Q_{\rm SP}$	$T_{\rm NET}$	σ_{SP}	τ_{SP}
$U_{\mathrm{FB}}~(\%\bar{D})$	0.7	7.7	9.8	0.1	0.7	2.9	0.9

 $P_{\rm D}$ is obtained from the measured revolutions per second (n) and toque (τ)

$$P_{\rm D} = 2\pi \times n \times \tau. \tag{10}$$

The theoretical framework proposed by the specialist committee was standardized and validated by means of a rigorous experimental campaign from seven institutes on an Athena model at Fr = 0.6 [32]. Subsequently, NSWCCD conducted extensive WJ powering experiments the JHSS model equipped with four adjacent waterjets [15].

3.3. DC model test results

Quantitative estimation of facility bias [25] requires a minimum of three facilities. Here, data (D) is available from two facilities, and a qualitative estimate for facility bias (U_{FB}) can be obtained as a percent of the mean data (\overline{D}).

$$U_{\rm FB} = \left| \frac{D_{\rm INSEAN} - D_{\rm BSHC}}{2} \right| \% \bar{D}.$$
 (11)

Figure 5 shows the data from the flow rate measurements of the bollard pull tests and the velocity profile measurements at stations 1 and 6, used for the momentumflux analysis. Figure 6 compares R_{BH} , T_{NET} , Q_{SP} , and the dynamometer shaft thrust (T_S) and Fig. 7 compares σ and τ for both BH (hollow symbols) and SP (filled symbols) conditions. For the towed BH tests, the facilities show very good agreement of all data up to Fr = 0.5. U_{FB} for R_{BH} progressively increases from 1.5% at Fr = 0.5, to 3.5% at Fr = 0.7. U_{FB} for sinkage σ_{BH} increases with increasing Fr, and the resulting variation in dynamic wetted area accounts for larger U_{FB} for R_{BH} at higher Fr. Though the facilities have different vertical pivots, trim τ_{BH} shows good agreement over the entire Fr range with average $U_{FB} = 9.8\%$. For Fr < 0.4, \overline{D} for τ_{BH} approach zero resulting in large U_{FB} values.

For SP measurements, Q_{SP} shows very good agreement over the Fr range with average $U_{FB} = 0.1\%$. Table 2 provides a summary of the U_{FB} values for the BH and SP measurements.

Figure 8 shows the comparison of the decomposed system efficiencies. Accurate calculation of P_{PE} requires measurement of pressure head at stations 3 and 5, which is challenging and expensive. Instead, both facilities used the product of T_{S} and V_{Ship} as an approximate measure for P_{PE} . Since energy fluxes at stations 1 and 7 were



Fig. 5. Flow rate and velocity profiles: (a) Bollard pull flow rate measurements, (b) Velocity profile at station 1, (c) Velocity profile at station 6. (Colors are visible in the online version of the article; http://dx. doi.org/10.3233/ISP-130098.)

not calculated, P_{JSE} is unavailable and η_{duct} and η_{jet} are not decomposed. $\eta_{\text{duct}} \times \eta_{\text{jet}} = P_{\text{TE}}/P_{\text{PE}}$ is reported instead. U_{FB} values for the decomposed efficiencies are tabulated in Table 3.

3.4. Analysis of experimental results

3.4.1. Performance analysis of DC WJ

An indication of the *DC WJ* design performance is obtained by comparing its efficiencies with JHSS (Table 4) which serves as a benchmark [15]. All system efficiencies for DC are smaller than JHSS.

JHSS has a larger η_{pump} since the experiments used full-scale thrust loading similarity with the incorporation of an added tow force. The excess energy of the ingested working fluid due to the added tow force creates a higher ram pressure at the impeller





Fig. 6. Powering performance data. (Colors are visible in the online version of the article; http://dx.doi.org/ 10.3233/ISP-130098.)



Fig. 7. Sinkage and trim data. (Colors are visible in the online version of the article; http://dx.doi.org/ 10.3233/ISP-130098.)

plane resulting in better pump performance. Without an added tow force, the DC requires larger Q_{SP} and n and the pump operates at a larger specific speed resulting in a reduced efficiency.

This disparity also accounts for the difference in thrust deduction factor (1 - t) as the larger $Q_{\rm SP}$ causes larger suction pressure at the inlet, and hence a greater increase in $\sigma_{\rm SP}$ and $\tau_{\rm SP}$ resulting in larger $T_{\rm NET}$.



Fig. 8. Decomposed system efficiencies. (Colors are visible in the online version of the article; http://dx. doi.org/10.3233/ISP-130098.)

Table 3						
Decomposed efficiencies U_{FB}						
	$\eta_{\rm D}$	η_{pump}	$\eta_{\rm duct} imes \eta_{ m jet}$	1 - t		
U_{FB} (% \bar{D})	2.1	0.5	1.3	1.2		

Table 4 Decomposed efficiencies comparison					
	$\eta_{\mathrm{D}}(\%)$	η _{pump} (%)	$\eta_{\text{duct}} \times \eta_{\text{jet}}$ (%)	1 - t(%)	
DC	28	45	66	85	
JHSS	45	58	85	91	

The jet velocity ratio $JVR = V_{\text{ship}}/V_{\text{jet}}$ and $IVR = V_{\text{ship}}/V_{\text{pump}}$ influence η_{jet} and η_{duct} values, respectively. Note, to keep JVR and IVR values bounded when V_{ship} goes to zero, the definitions used here follow Bulten [3], which is the reciprocal of that used in some literature.

JVR for *DC* and JHSS are 0.60 and 0.66, respectively. For optimal η_{jet} , *JVR* values should be in the range of 0.65 and 0.75 [3]. Lower *JVR* values for *DC* result in excess axial kinetic energy loss into the wake, thereby reducing η_{jet} .

IVR for *DC* and JHSS are 1.83 and 1.7, respectively. Waterjets with *IVR* values greater than 1.8 have increased risk of separation at the top side of the inlet due to sudden flow deceleration [3], thereby reducing η_{duct} . *CFD* simulations confirmed the occurrence of flow separation for *DC*, making the inlet a good candidate for optimization.

M. Kandasamy et al. /	Simulation based	design	optimization
2			1 2

		Table 5					
Comparison of U_{FB}							
	T_{NET} (%)	Q_{SP} (%)	$\sigma_{\mathrm{SP}}(\%)$	$ au_{\mathrm{SP}}(\%)$			
Athena	± 18	± 5	±116	± 27			
DC	± 0.7	± 0.1	± 2.9	± 0.9			

3.4.2. Analysis of facility bias

The ITTC [13] standardized experimental campaign on the Athena model, which was conducted by seven facilities, provides a basis of comparison for U_{FB} . The values are compared in Table 5, and U_{FB} for *DC* are significantly smaller. Both BSHC and INSEAN participated in the experimental campaign on the Athena model, but have had limited experience with the *ITTC WJ* model since that time.

4. Verification and validation

Qualitative validations of the HF and LF solutions were performed for both BH and SP cases over the Fr range. Detailed quantitative verification and validation was performed for Fr = 0.5.

4.1. Overview of V&V methodology

Verification and Validation procedures follow Stern et al. [27]. Verification procedures estimate numerical uncertainties (U_{SN}) based on iterative (U_I) and grid (U_G) uncertainties

$$U_{\rm SN} = \sqrt{U_{\rm I}^2 + U_{\rm G}^2}.$$
 (12)

Grid convergence studies are carried out for three solutions (S) with systematic refinement ratio $r = \frac{\Delta x_2}{\Delta x_1} = \frac{\Delta x_3}{\Delta x_2}$, where 3, 2 and 1 represent the coarse, medium, and fine grids, respectively. Solution changes ε and the convergence ratio R are defined as $\varepsilon_{ij} = S_i - S_j$ and $R = \frac{\varepsilon_{12}}{\varepsilon_{23}}$. For monotonic convergence, 0 < R < 1, factor of safety method [34] is used for estimations of $U_{\rm G}$. The ratio $P = \frac{p_{\rm RE}}{p_{th}}$ is used to estimate the factor of safety and $U_{\rm G}$ is given by

$$U_{\rm G} = \begin{cases} (2.45 - 0.85P) |\delta_{\rm RE}|, & 0 < P \leq 1, \\ (16.4P - 14.8) |\delta_{\rm RE}|, & 1 < P, \end{cases}$$
(13)

where,

$$\delta_{\rm RE} = \frac{\varepsilon_{21}}{r^{p_{\rm RE}} - 1}.\tag{14}$$

For oscillatory convergence, -1 < R < 0, U_G is estimated from the upper and lower bounds of the oscillation. U_G is undefined for monotonic divergence, R > 1, and oscillatory divergence, R < -1.

Validation procedure defines the comparison error (E) and the validation uncertainty (U_V) using experimental benchmark data (D) and its uncertainty (U_D) . If U_V bounds E, the combination of all the errors in D and S is smaller than U_V and validation is achieved at the U_V interval, where

$$\mathbf{E} = \mathbf{D} - \mathbf{S},\tag{15}$$

$$U_{\rm V} = \sqrt{U_{\rm D}^2 + U_{\rm SN}^2}.$$
 (16)

4.2. HF and LF models

Table 6 provides the WJ induced vertical forces (C_{Tz}) and moments (M_{Ty}) about the centre of gravity, non-dimensionalized by the V_{ship} and LWL.

Figure 9 shows the overset WJ grid used for HF simulations of duct flow using the actuator disk model. T_{net} and P_{PE} vary relative to the square and cube of Q_{SP} , respectively, and hence accurate prediction of Q_{SP} is vital. For the current study, the duct was discretized using a single structured grid, which overlaps with the hull grid

 Table 6

 WJ induced forces and moments

 $\overline{\mathrm{Fr}}$ C_{Tz} M_{Ty}

 0.3 -0.03×10^{-4} 0.02×10^{-4}

 0.5 -3.93×10^{-4} 1.61×10^{-4}

 0.7 -1.35×10^{-4} 0.82×10^{-4}



Fig. 9. Overset grid for duct discretization. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)

Table 7						
Multi-block grid densities						
Block #	Description	# grid pts				
1	Inner-hull	3,540,908				
2	Outer-hull	3,540,908				
3	Duct	4,099,579				
4	Refinement	2,285,061				
5	Background	4,355,778				
	Total	17,822,233				

at the inlet and the nozzle exit since additional overset grids inside the duct cause interpolation errors as seen in Takai et al. [31].

Table 7 shows the fine grid (S₁) densities for the overset blocks used for the HF simulations. The HF and LF solvers used systematic refinement ratios $r = \sqrt{2}$ and 2, respectively.

For LF, the same number of grid panels has been used for hull and free surface: 6000 on S_1 , 3000 on S_2 and 1500 on S_3 , for a total number of panels of 12,000 on S_1 , 6000 on S_2 and 3000 on S_3 . For HF simulations the grids densities for S_1 , S_2 and S_3 are 17.8M, 6.3M and 2.3M, respectively.

4.3. Qualitative validation over Fr range

HF solutions using the medium grid were obtained at Fr = 0.3, 0.5 and 0.7 for both BH and SP simulations. LF solutions were obtained over the Fr range for BH simulations, and at Fr = 0.5 using the simplified *CFD WJ* model. The solutions are compared with \overline{D} over the Fr range in Fig. 10.

For BH simulations, $R_{\rm BH}$ for both HF and LF calculations agree well with \bar{D} . σ for LF calculations are under-predicted for Fr < 0.5 and over-predicted for Fr > 0.5 and τ calculations show a reversed trend. $\sigma_{\rm BH}$ for HF calculations are under-predicted for all values and $\tau_{\rm BH}$ is under-predicted at Fr = 0.5.

For SP simulations, both HF and LF calculations under-predict T_{NET} , Q_{SP} and σ_{SP} for Fr ≥ 0.5 . Data is unavailable for Fr = 0.3. τ_{SP} is over-predicted by HF and under-predicted by LF.

4.4. Quantitative V&V at Fr = 0.5

At Fr = 0.5, a grid verification and validation study was conducted for both barehull and self-propelled simulations. $R_{\rm BH}$, $\sigma_{\rm BH}$ and $\tau_{\rm BH}$ for both LF and HF simulations are tabulated in Table 8. $U_{\rm I}$ is negligible and $U_{\rm SN} = U_{\rm G}$. Both HF and LF solutions show monotonic grid convergence for all the quantities. For all cases, 0 < R < 1, and monotonic convergence was achieved. For HF calculations P < 2and reasonably close to 1. For LF calculations P > 3, which make the $U_{\rm SN}$ calculations unreliable since the available database used by Xing and Stern [34] for the development of the factor of safety method was restricted to P < 2.



Fig. 10. Qualitative V&V. (Colors are visible in the online version of the article; http://dx.doi.org/ $10.3233/ISP{-}130098.)$

Table 8									
	BH verification of HF and LF solutions								
	S ₃	S ₂	S ₁	R	P	U_{SN}			
	HF BH								
$R_{\rm BH}$	84.75	86.03	86.52	0.38	1.39	2.78			
$\sigma_{\rm BH}$	3.173	3.188	3.192	0.29	1.82	0.17			
$ au_{ m BH}$	1.569	1.669	1.712	0.43	1.22	9.79			
		Ι	F BH						
$R_{\rm BH}$	86.70	87.97	88.09	0.12	3.00	0.26			
$\sigma_{ m BH}$	4.129	4.188	4.191	0.50	6.92	0.05			
$ au_{ m BH}$	1.754	1.790	1.794	0.11	6.34	0.04			

295

-483-

Table 9 BH validation of HE and LE solutions							
	D	U _{FB}			E		
			HF	LF	HF	LF	
$R_{\rm BH}$	89.06	3.00	3.85	3.01	0.55	-1.09	
$\sigma_{ m BH}$	3.182	1.31	1.31	1.31	0.20	31.57	
$ au_{ m BH}$	1.751	1.00	9.84	1.00	2.33	2.63	

		Ta	ble 10							
	SP verification of RANS and LF solutions									
	S ₃	S ₂	S ₁	R	P	U_{SN}				
	HF SP									
Q_{SP}	1.860	1.897	1.915	0.49	1.04	2.15				
T_{NET}	96.16	102.02	104.22	0.38	1.41	2.91				
$\sigma_{ m SP}$	4.731	4.855	4.762	-	-	1.29				
$ au_{\mathrm{SP}}$	2.763	2.585	2.687	-	-	3.64				
P_{PE}	194.87	201.12	204.02	0.46	1.11	4.16				
		L	F SP							
T_{NET}	93.13	94.39	95.02	0.45	2.29	0.7				
$\sigma_{ m SP}$	4.845	4.827	4.883	-	-	NA				
τ_{SP}	2.081	2.123	2.140	0.49	2.08	2.14				

Since precision and bias errors were not quantified in the experiments, U_{FB} was used for validation. D_{INSEAN} was used for calculating E, since the optimized hull will also be tested at that facility. Table 9 shows the results of the validation studies. Except for LF σ , whose |E| lies outside the U_{V} interval, all other solutions are validated.

For SP simulations, solutions HF calculations were verified and validated for Q_{SP} , T_{NET} , P_{PE} , σ_{SP} and τ_{SP} . P_{PE} was obtained by calculating energy difference between stations 5 and 3. LF calculations were verified and validated for T_{NET} , σ_{SP} and τ_{SP} . The HF and LF solutions are tabulated in Table 10. U_{I} is negligible and $U_{\text{SN}} = U_{\text{G}}$. HF solutions show monotonic grid convergence for Q_{SP} , T_{NET} and P_{PE} , and oscillatory convergence for σ_{SP} and τ_{SP} . LF solutions show monotonic grid convergence for σ_{SP} .

Table 11 shows the validation results for the different quantities. HF solutions |E| lie outside the U_V interval for P_{PE} . Note that the P_{PE} estimate from the experiments does not account for viscous energy losses between station 3 and 5, and hence is prone to over-estimation. LF solutions |E| lie outside the U_V interval for T_{NET} .

The numerical uncertainty levels are similar to than obtained for JHSS [31]. The validation errors are smaller than that for JHSS, due to the elimination of overset grids within the duct. For DC, |E| = 2% and 4% for Q_{SP} , and T_{NET} , respectively, compared to 5.6% and 6.5% for JHSS.

Table 11							
SP validation of RANS and LF solutions							
	D	$U_{\rm FB}$	$U_{\mathbf{V}}$		U _V E		Е
			HF	LF	HF	LF	
$Q_{\rm SP}$	1.95	0.51	2.21	NA	-2.05	NA	
T_{NET}	106.9	3.21	4.35	3.28	-4.17	-11.1	
$\sigma_{ m BH}$	4.87	1.34	3.58	NA	-2.30	NA	
$ au_{ m BH}$	1.75	2.91	2.47	3.61	-2.33	2.63	
P_{PE}	232	8.20	9.22	NA	-12.06	NA	

5. Single objective resistance optimization using model

To facilitate greater variability of the design space, IIHR, DTMB and INSEAN conducted initial geometry sensitivity studies with the *CFD WJ* model to determine a feasible design space using different approaches. Multiple geometries were obtained using different geometry modification techniques; B-spline, Free Form Deformation (FFD), and CREATE-SHAPE [33], with resistance reductions varying from 0.5% to 1.5% compared to the original geometry. A morphing method, which enables direct construction of the design space by integrating the best geometries from the different sensitivity studies, was used for the optimization. The overall optimization process used a four-pronged approach by IIHR, DTMB, NMRI and INSEAN:

- IIHR and DTMB performed a single objective PSO optimization for resistance at Fr = 0.5 using different combinations of the initial geometries to explore different subset design spaces.
- INSEAN performed single objective PSO optimization for resistance at Fr = 0.5 using both variable fidelity and high fidelity optimization using generalized FFD with PSO optimizer to investigate computational cost reduction.
- NMRI performed MOGA for resistance at three speeds: Fr = 0.3, 0.5 and 0.7 [30].

The best BH geometries obtained for Fr = 0.5 from the different optimization approaches were then verified using *RANS* with identical grid size, grid topology and solver convergence criteria. The best geometry (Fig. 11) showed a resistance reduction of 4% due to significant reduction of the interference region trough (Fig. 12) and was selected for subsequent *WJ* inlet optimization.

6. Design optimization for overall propulsive efficiency of propelled hull

Figure 13 shows the results from the sensitivity analysis performed on the WJ inlet shape. The streamlines and C_P contours at the WJ inlet symmetry plane for the original geometry and the modified geometry are illustrated. Sensitivity analysis

M. Kandasamy et al. / Simulation based design optimization



Fig. 11. Comparison of the original (red) and optimized (green) starboard demi-hulls. (The colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)



Fig. 12. Wave elevation comparison. (Colors are visible in the online version of the article; http://dx. doi.org/10.3233/ISP-130098.)

on the transition angle, inlet-angle and ramp-radius showed that a reduction of the angles, combined with an increase in the ramp radius with a widened inlet rectified the flow separation and increased η_{duct} .

Figure 14 illustrates the inlet ramp design variations intended for smoother transition and increasing boundary layer ingestion for recovery of residual kinetic energy lost to the wake.

The PSO optimization was performed on a coarse grid by morphing three initial geometries constructed with the three combinations of the design variables. The flow solver was tuned for trend identification with a liberal convergence criterion to increase computational speed. The optimal design indicated ~10% reduction in $P_{\rm PE}$. However, a grid refinement analysis of the optimized geometry revealed unforeseen problems: the fine grid solution predicted deeper trough at the stern with the free surface very close to the inlet, which was judged a possible cause of air ingestion into the *WJ* inlet. Therefore, local modifications on the inner side of the demi-hull – close to the *WJ*-inlet – were made (Fig. 15) and a geometrically constraint-based optimization was performed. The constraint was defined as the minimum distance function from the free surface to the inlet to be greater than or equal to the original Delft catamaran. Since the geometry variations were localized, the speed of the





Fig. 13. Design variable sensitivity analysis: (a) original geometry, (b) modified geometry. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)



Fig. 14. Inlet ramp design variation: (a) original geometry, (b) modified geometry. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)



Fig. 15. Localized inlet design variations, green shade is the initial optimized geometry, grey shade is the modified design. (The colors are visible in the online version of the article; http://dx.doi.org/10.3233/ ISP-130098.)

M. Kandasamy et al. / Simulation based design optimization



Fig. 16. Localized inlet design variations to minimize possibility of air entrainment. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)



Fig. 17. Original and optimized waterjets. (Colors are visible in the online version of the article; http://dx. doi.org/10.3233/ISP-130098.)

fine grid computations was increased by using restart solution files from comparable geometries. The solutions for the final optimized geometry is shown in Fig. 16 compared to the initially optimized geometry.

Comparison of the duct surface pressure contours on the original and optimized hulls (Fig. 17) illustrates the increase in ram pressure on the forward facing walls of the optimized duct. This added potential energy develops additional pressure at the nozzle, which converts it to kinetic energy at exit and increases the *WJ* efficiency.

Detailed performance comparisons are tabulated in Table 12. The optimized hull shows 11% decrease in powering requirement. The *WJ* system efficiencies for the duct, and thrust deduction are improved by 6.7% and 1.32%, respectively. The jet efficiency shows a modest improvement of 0.1%.

Grid verification studies (Table 13) were conducted for the optimized geometry, using both barehull and self-propelled simulations. $U_{\rm I}$ is negligible and $U_{\rm SN} = U_{\rm G}$. For all cases, 0 < R < 1, and monotonic convergence was achieved. P < 2 and reasonably close to 1. The $U_{\rm SN}$ values are similar to that obtained for the original geometry.

Previous optimization of the JHSS waterjet inlet curvature using the same methodology yielded a just 2% decrease in powering requirement. However, a drastic design modification by merging the adjacent inlets (Fig. 18) yielded an 8% decrease in powering requirement.

Table 12									
	Performance comparison								
Ori. Opt. Opt%Or									
	Resistance								
$R_{\rm BH}$	4.32E+01	4.15E+01	-3.94						
T_{NET}	5.21E+01	4.94E+01	-5.18						
	Pov	vering							
P_{E}	1.29E+02	1.24E+02	-3.94						
P_{TE}	1.55E+02	1.47E+02	-5.18						
P_{JSE}	1.80E+02	1.71E+02	-5.28						
P_{PE}	2.04E+02	1.81E+02	-11.22						
Waterjet system efficiencies									
$\eta_{ m duct}$	8.82E-01	9.41E-01	+6.70						
η_{jet}	8.63E-01	8.64E-01	+0.10						
(1 - t)	8.29E-01	8.40E-01	+1.32						

Table 13

	S ₃	S ₂	S ₁	R	P	$U_{\rm SN}$				
BH with CFD WJ model										
$R_{\rm BH}$	81.05	82.52	83.11	0.40	1.32	3.24				
$\sigma_{ m BH}$	3.164	3.193	3.205	0.41	1.27	0.47				
$ au_{\mathrm{BH}}$	1.504	1.615	1.667	0.47	1.09	8.64				
	SP with discretized duct									
$Q_{\rm SP}$	1.832	1.861	1.874	0.45	1.16	2.36				
$T_{\rm NET}$	92.14	97.05	98.83	0.36	1.46	2.51				
$\sigma_{ m BH}$	4.578	4.727	4.765	0.26	1.97	4.79				
$ au_{ m BH}$	2.511	2.632	2.677	0.37	1.43	8.56				
P_{PE}	171.68	178.84	181.12	0.32	1.65	7.22				

7. Multi-objective optimization using

The definition of the multi-objective optimization problem is based on the selection of three objective functions, that is, the total resistance at the speed of Fr = 0.3, 0.5 and 0.7. A subset of the previous base hulls has been adopted, in order to reduce the overall dimension of the design space and the complexity of the optimization problem accordingly. Since the objective functions are now three, the overall computational cost is triple, since three distinct runs of the HF solver are needed in order to produce the complete evaluation of one position in the design variable space.

model

For this multi-objective problem, the final solution of the problem is not represented by a single geometry, but by a suite of different solutions, representing the Pareto optimal set. In the case of three objective functions, the Pareto optimal set

M. Kandasamy et al. / Simulation based design optimization



Fig. 18. Optimization of JHSS waterjet by merging the inlets. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)

is represented by a 3D surface, and its representation and visualization is not easy. Consequently, here are reported three different views, orthogonal to each coordinate axis related to an objective function. In each graph, the variety along the missing axis is lost, but we can observe to position of some interesting solutions and their performances in comparison with all the other solutions.

In Fig. 19, the three different views are reported. Red dot is indicating the best solution for function 1 (total resistance at Fr = 0.3), green dot is indicating the best solution for function 2 (total resistance at Fr = 0.5), and blue dot is indicating the best solution for function 3 (total resistance at Fr = 0.7). It is evident how there is not improvement for the first objective function: this is probably connected to the fact that the base geometries are obtained with a particular emphasis to the central speed Fr = 0.5. Looking at the second view, reporting second and third objective functions, a correlation is nearly evident, also if there are two distinct solutions representing the best for each objective function. On the contrary, the cloud of points in the space of the first and second objective function, as well as for the space of the first and third, is not crossing the vertical axis, showing a negative correlation. This means that an improvement on one of the objective if resulting into a deterioration of the other objective, that is, the two objectives are in opposition each other, and it is not possible to improve them together. A different parameterization scheme would be probably able to provide improved shapes also for the lower speed, while it is not possible to argue if the complete opposition between the objective functions is solvable by changing the parameterization or not.

In Fig. 20, a comparison between the section views for the original DC and one of the Pareto optimal solutions is reported, together with a comparison of the sectional area curve distributions. It is possible to clearly observe how there is a shift of the volume from inner to outer, and from stern to bow.



Fig. 19. Different views of the Pareto front for the multi-objective optimization problem. (Colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)



M. Kandasamy et al. / Simulation based design optimization

Fig. 20. Comparison between the original shape and the shape of best compromise solution for the threeobjectives optimization problem. (Colors are visible in the online version of the article; http://dx.doi.org/ 10.3233/ISP-130098.)

8. Variable fidelity optimization for computational efficiency as proof of concept

The overall goal of the VF/VP approach is to obtain a significant CPU time reduction while at the same time, regain the same optimal solution as if we were solving the full-HF problem. Therefore, to assess the success of the VF/VP algorithm, the same optimization problem (same parameterization, constraints and objective function) for the DC has been carried out twice: first using the HF alone and then solving again the same problem using the VF/VP algorithm.

The parameterization scheme used is this test is the FFD approach. A single FFD box surrounds the hull: 4 parameters are used to shift the hull sections sideways, one is used to move the sections longitudinally and one is used to change the transom stern depth, for an overall number of 6 design variables.



Fig. 21. Transversal sections of the original (black) and optimized hulls (red), both full HF and VF/VP solutions. (The colors are visible in the online version of the article; http://dx.doi.org/10.3233/ISP-130098.)

The shape of optimal design obtained by the HF and by VF/VP is substantially the same. The two geometries are superimposed in Fig. 21. In terms of computed objective function, the difference between full HF and VF/VP is of about 0.05% (6.4412 for the full HF, 6.4447 for the VF/VP). A great reduction is obtained instead in the computational cost: the HF optimization has required 940 HF evaluations of the objective function, while the VF/VP only needed 532 HF computations, plus 940 (almost inexpensive compared to HF SMD) LF evaluations. The overall CPU time reduction is therefore about 54%.

9. Conclusions

A simulation-based design optimization for the hull and WJ inlet was carried out for the powering optimization of WJ propelled DC, using integrated computational and experimental fluid dynamics. A WJ equipped DC was constructed and tested at two facilities with good agreement of data. The data was used to validate the LF and HF solvers and for the formulation of a simplified *CFD* WJ model that was used in conjunction with the LF optimization.

The particle swarm optimizer was used for single speed optimization at Fr = 0.5, and genetic algorithms were used for multi speed optimization at Fr = 0.3, 0.5 and 0.7. The multi-speed optimization showed design improvement at Fr = 0.5 and 0.7, but not at Fr = 0.3 since the design variables were obtained with a particular emphasis to the higher speeds. High fidelity simulation results for the optimized barehull geometry at Fr = 0.5 indicated 4% reduction in resistance and the optimized *WJ* equipped geometry indicated 11% reduction in effective pump power required at self-propulsion.

Ongoing developments for geometric variability exploration, based on Karhunen– Loève expansion (*KLE*), have shown the capability of producing a wider range of design possibilities with deeper improvements [10,12]. Accordingly, a new optimization campaign [8] will be performed using reduced dimensional research spaces as provided by *KLE* analysis. Meta-models and/or *VF/VP* approaches will be used to reduce the computational time.

Best design overall will be built and tested at INSEAN using the same experimental set-up as was used for the original model, to reduce the comparison errors. A complete set of tests is planned in the near future.

Future direction for design optimization is to include uncertainty effects on objectives and constraints. Ship designers have been always concerned with the uncertainties of the environment in which the ship sails (waves, winds, currents) since mostly not avoidable and often responsible for performance loss and failures. Robust and reliability-based design optimization methods, developed to improve product quality and reliability in industrial engineering, are to prevent performance drop when operating in off-design conditions and avoid dramatic failures in the case of exceptional events. The Bayesian approach is used to formulate the problems of robust design optimization (RDO) and reliability-based design optimization (RBDO). These require the uncertainty quantification (UQ) of the relevant simulation outputs over the stochastic inputs domain. The difficulty with exploiting this framework is mainly computational, since UQ requires the numerical integration of expensive simulation outputs over the uncertainties involved. Accordingly, research in UQ is an important precursory step for RDO/RBDO providing the impact of stochastic inputs on relevant outputs and identifying the most efficient UQ methods (such as meta-models based analyses) for the problem addressed. Earlier and current UQ research includes development and assessment of a framework for convergence, validation, and comparison with deterministic V&V of UQ studies. Applications include NACA 0012 hydrofoil with variable Re [22]; UQ of DC calm water resistance, sinkage and trim with variable Fr and geometry [10]; and UQ for DC resistance, motions and slamming loads in stochastic wave and variable geometry [12]. Future RDO/RBDO activities will focus on DC optimal design for reduced resistance, motions and slamming loads in stochastic wave at sea state 7 [11]. Geometric variability will be explored using KLE and, in order to keep the computational effort reasonable, optimization and UQ will be performed using metamodels and/or VF/VP methods.

Savings obtained are demonstrating the usefulness the VF/VP formulation. For this reason, further verification of the VF/VP approach will be also carried out in the near future, in order to gain more insight about the potential of this technique. Different formulations of the scheme, including the use of the variance estimation provided by the kriging meta-model in order to adapt the "trust region" radius dynamically will be explored.

Acknowledgements

The present research is supported by the Office of Naval Research, Grant N00014-08-1-0957, under the administration of Dr. Ki-Han Kim, and NICOP Grant N00014-01-543 under the administration of Dr. Patrick Purtell. The URANS computations were performed at the NAVY DoD Supercomputing Resource Center. The authors thank Thad Michael and Donnelly Martin of NSWCCD for their assistance.
References

- N.M. Alexandrova and R. Lewis, First-order approximation and model management in optimization, in: *Large-Scale PDE Constrained Optimization*, L.T. Biegler, O. Ghattas, M. Heinkenschloss and B. van Bloemen Waanders, eds, Lecture Notes in Computational Science and Engineering, Springer, Berlin/Heidelberg, 2003.
- [2] P. Bassanini, U. Bulgarelli, E.F. Campana and F. Lalli, The wave resistance problem in a boundary integral formulation, *Surveys on Mathematics for Industry* 4 (1994), 151–194.
- [3] N.W.H. Bulten, Numerical analysis of a WJ propulsion system, PhD thesis, University of Eindhoven, 2006.
- [4] E.F. Campana, D. Peri, A. Pinto, Y. Tahara and F. Stern, Shape optimization in ship hydrodynamics using computational fluid dynamics, *Computer Methods in Applied Mechanics and Engineering* 196 (2006), 634–651.
- [5] E.F. Campana, D. Peri, M. Kandasamy, Y. Tahara and F. Stern, Numerical optimization methods for ship hydrodynamic design, in: SNAME Annual Meeting, Providence, RI, USA, 2009.
- [6] P.M. Carrica, R.V. Wilson, R.W. Noack and F. Stern, Ship motions using single-phase level set with dynamic overset grids, *Computer and Fluids* 36 (2007), 1415–1433.
- [7] T. Castiglione, F. Stern, S. Bova and M. Kandasamy, Numerical investigation of the seakeeping behavior of a catamaran advancing in regular head waves, *Ocean Engineering* 38(16) (2011), 1806–1822.
- [8] X. Chen, M. Diez, M. Kandasamy, E.F. Campana and F. Stern, Optimization of waterjet propelled Delft-catamaran in calm water using Karhunen–Loève expansion and meta-heuristics, abstract submitted for presentation at *12th International Conference on Fast Sea Transportation*, FAST 2013, Amsterdam, The Netherlands, September 2013.
- [9] K. Delaney, M. Donnely, M. Elbert and D. Fry, Use of RANS for waterjet analysis of a high-speed sealift concept vessel, in: 1st International Symposium on Marine Propulsors, Trondheim, Norway, 2009.
- [10] M. Diez, W. He, E.F. Campana and F. Stern, Uncertainty quantification of Delft catamaran resistance, sinkage and trim for variable Froude number and geometry using metamodels, quadrature and Karhunen–Loève expansion, *Journal of Marine Science and Technology* (2013), to appear.
- [11] M. Diez, E.F. Campana and F. Stern, Reliability-based robust design optimization of Delft catamaran in stochastic wave, abstract submitted for presentation at 12th International Conference on Fast Sea Transportation, FAST 2013, Amsterdam, The Netherlands, September 2013.
- [12] W. He, M. Diez, E.F. Campana and F. Stern, URANS study of Delft catamaran total/added resistance, motions and slamming loads in head sea including irregular wave and uncertainty quantification for variable regular wave and geometry, *Ocean Engineering* (2013), to appear.
- [13] ITTC Propulsion Committee, Testing and extrapolation methods, high speed marine vehicles, waterjets, propulsive performance prediction, ITTC-Recommended Procedures and Guidelines, 7.5-02-05-03.1, 2005.
- [14] C. Janson and L. Larsson, A method for the optimization of ship hulls from a resistance point of view, in: 21st Symposium on Naval Hydrodynamics, Norway, 1996.
- [15] S. Jessup, M. Donnelly, D. Fry, D. Cusanelli and M. Wilson, Performance analysis of a four waterjet propulsion system for a large sealift ship, in: 27th Symposium on Naval Hydrodynamics, Seoul, Korea, 2008.
- [16] M. Kandasamy, S.K. Ooi, P. Carrica and F. Stern, Integral force/moment waterjet model for CFD simulations, Journal of Fluid Engineering 132(10) (2010), 101103: 1–9.
- [17] M. Kandasamy, S.K. Ooi, P. Carrica, F. Stern, E. Campana, D. Peri, P. Osborne, J. Cote, N. Macdonald and N.D. Waal, Multi-fidelity optimization of a high speed foil-assisted semi-planing catamaran for low wake, *Journal of Marine Science and Technology* 16(2) (2011), 143–156.

- [18] M. Kandasamy, W. He, T. Takai, Y. Tahara, D. Peri, E.F. Campana, W. Wilson and F. Stern, Optimization of WJ propelled high speed ships – JHSS and Delft catamaran, in: *Proceedings 11th International Conference on Fast Sea Transportation, FAST 2011*, Honolulu, HI, USA, 2011.
- [19] E. Milanov, V. Chotukova and F. Stern, Experimental and simulation studies on fast Delft372 catamaran maneuvering and course stability in deep and shallow water, in: *Proceedings 11th International Conference on Fast Sea Transportation, FAST 2011*, Honolulu, HI, USA, 2011.
- [20] E. Milanov, Model tests of waterjet propelled Delft 372 catamaran, BSHC Report KP092006/01, December 2010.
- [21] M. Miozzi, Model tests of waterjet propelled Delft 372 catamaran, INSEAN-CNR internal report 2011-TR-011, 2011.
- [22] S.M. Mousaviraad, W. He, M. Diez and F. Stern, Framework for convergence and validation of stochastic uncertainty quantification and relationship to deterministic verification and validation, *International Journal for Uncertainty Quantification* 3(5) (2013), 371–395.
- [23] D. Peri, Self-learning metamodels for optimization, Ship Technology Research 56 (2009), 94-108.
- [24] D. Peri, E.F. Campana, Y. Tahara, T. Takai, M. Kandasamy and F. Stern, New developments in simulation-based design with application to high speed water-jet ship design, in: 28th Symposium on Naval Hydro-Dynamics, Pasadena, CA, USA, 2010.
- [25] F. Stern, A. Olivieri, J. Shao, J. Longo and T. Ratcliffe, Statistical approach for estimating intervals of certification or biases of facilities or measurement systems including uncertainties, *J. Fluids Eng.* 127 (2005), 604–610.
- [26] F. Stern, P. Carrica, M. Kandasamy et al., Computational hydrodynamic tools for high-speed sealift, *Transactions SNAME* 114 (2006), 55–81.
- [27] F. Stern, R. Wilson and J. Shao, Quantitative approach to V&V of CFD simulations and certification of CFD codes with examples, *International Journal Numerical Methods Fluids* 50(11) (2006), 1335–1355, Special Issue: Advances in Computational Heat Transfer.
- [28] F. Stern, P. Carrica, M. Kandasamy et al., Computational hydrodynamic tools for high speed sealift: Phase II final report, IIHR Technical Report No. 465, 2008.
- [29] Y. Tahara, D. Peri, E.F. Campana and F. Stern, *CFD*-based multi-objective optimization of a surface combatant by using global optimization method, *Journal of Marine Science and Technology* 13 (2008), 95–116.
- [30] Y. Tahara, T. Hino, M. Kandasamy, W. He and F. Stern, CFD-based multiobjective optimization of waterjet propelled high speed ships, in: Proceedings 11th International Conference on Fast Sea Transportation, FAST 2011, Honolulu, HI, USA, 2011.
- [31] T. Takai, M. Kandasamy and F. Stern, Verification and validation study of URANS simulations for an axial WJ propelled large high-speed ship, *Journal of Marine Science and Technology* 16(4) (2011), 434–447.
- [32] T. Van Terwisga (Chairman), Report of the specialist committee on validation of waterjet test procedures, in: *Proceedings 24th International Towing Tank Conference II*, 2005, pp. 471–508.
- [33] W. Wilson, D. Hendrix and J. Gorski, Hull form optimization for early stage ship design, Naval Engineers Journal 122(2) (2010), 53–65.
- [34] T. Xing and F. Stern, Factors of safety for Richardson extrapolation, ASME J. Fluids Eng. 132 (2010), 061403: 1–13.

308

Sail Performance Analysis of Sailing Yachts by Numerical Calculations and Experiments

Y. Tahara, Y. Masuyama, T. Fukasawa and M. Katori National Maritime Research Institute, Kanazawa Institute of Technology Osaka Prefecture University, North Sails Japan

1. Introduction

Sails of a sailing yacht can be considered as multiple soft thin wings (membrane wings) with relative large cambers, and are often used at large attack angles. The shape of sail is determined as an equilibrium state of both aerodynamic force and tension acting on the sail surface. In particular, a spinnaker used for the running condition is a very soft membrane like a parachute, and the shape is simply formed by self-generated aerodynamic forces which are strongly affected by the sail shape itself. These facts lead to new challenges in the present problem, i.e., in the measurements the sail shape must be accurately measured in the flying condition, and in numerical simulation of flow and forces the sail flying shape is correctly given or predicted as a part of solution. The present study concerns the authors' ongoing effort on analyses of sail performance of sailing yachts by numerical calculations and experiments, and in this paper, the focus of discussions is more on the former. Two computational fluid dynamics (CFD) methods are used in the present study, and the results are validated through detailed comparison with experimental data. The data are obtained in onboard full-scale measurements by using a sail dynamometer boat. Our study concerns both the upwind and downwind sailing conditions; however, we focus on the former in the present chapter due to the limitation of space in this book. More detailed background of the present work is well described in Masuyama et al. (2009).

One of the two CFD methods is a Vortex Lattice method (VLM). Although the VLM is a potential flow calculation, it is well known the results agree well with the measured data at the upwind condition of small attack angle. The VLM is used as the sail design and making tool due to the quick convergence ability for the parametric survey of sail shape to obtain the desired sail performance, and also due to good compatibility with the finite element method (FEM) for the strength analysis. In this paper, a method to shed wake vortices step-by-step developed by Fukasawa was adopted in the Vortex Lattice method (Fukasawa, 1993; Fukasawa & Katori, 1993).

Another CFD method is a Multiblock Reynolds-Averaged Navier-Stokes (RANS)-based CFD named "*FLOWPACK*". This code was developed by Tahara specifically for CFD education and research, and design applications for ship hydrodynamics, aerodynamics, and fluid engineering (Tahara, 2008). As part of the developments for application to design problems, a complete multiblock domain decomposition feature was included. The numerical method of *FLOWPACK* solves the unsteady RANS and continuity equations for mean velocity and pressure. Either a zero or a two-equation turbulence model can be used for turbulence flow calculation, and in the present study the former was used. The *FLOWPACK* was included as a

solver in a sail performance analyzer named "*Advanced Aero Flow* (*AAF*)" developed by Katori (Katori, 2009). The AAF is a specialized package for the calculation of sail performance of sailing yachts, and composed of both mesh generator and post analyzer.

The sail shapes and performance were measured using a sail dynamometer boat *Fujin* under sailing condition on the sea (Masuyama et al., 1997a, 1997b). *Fujin* is a 34-foot LOA boat, in which load cells and CCD cameras were installed to simultaneously measure the sail forces and shapes. At the same time, the sailing conditions of the boat, e.g., boat speed, heel angle, wind speed, and wind angle, were measured. The shapes and 3D coordinates of the sails were used for the input data of the numerical calculations, and the calculated results were compared with the measured data. The sail coordinates with aerodynamic coefficients are tabulated for some sailing conditions in order to provide benchmark data for the CFD validation.

In this paper, overview of the above-mentioned CFD methods and experiments are described. As the aforementioned, sail flying shapes are considered in the present CFD so that the accurate prediction of flow and aerodynamic forces is possible. Discussion of the results is based on detailed comparison with the measurements. The discussion also includes the current capability of the CFD methods in the present problem, and prognosis for the enhancement of the capability in future work for higher accuracy and/or more complicated flow simulation. It will be noteworthy that the overall trends of the flow and the aerodynamic forces measured in the experiments are fairly well predicted by the present computations; and at the same time, experimental techniques originally implemented and used in the present study are shown very promising and capable to provide very detailed benchmark data for CFD validation.

2. Sail plan for the analysis

In this study the experiments and numerical calculations were performed for the upwind sailing condition. The sail shapes and performance were measured using a sail dynamometer boat *Fujin*. The sail plan of the *Fujin* and the coordinate system are shown in Fig.1. The principal dimensions of the boat and the detailed measurements of the sails are also shown in Table 1. The measurement system of the boat and testing conditions are described in section 5, and the measured and calculated results are compared and discussed in section 6.



Fig. 1. Schematic showing the sail plan of Fujin with 130% jib and the coordinate system

HULL		SAIL DIMENSIONS						
Length Over All [m]	10.35		Mainsail	130% Jib				
Length Water Line [m]	8.80	Peak Height [m]	13.82	10.70				
Breadth Maximum [m]	3.37	Luff Length [m]	12.50	11.45				
Breadth Water Line [m]	2.64	Foot Length [m]	4.44	4.89				
Displacement [ton]	Sail Area [m ²]	33.20	26.10					
SAIL	Height [%]	Chord Length [m]						
I [m]	11.00	0	4.44	4.89				
J [m]	3.61	10	4.13	4.44				
P [m]	20	3.85	3.94					
E [m]	4.51	40	3.23	2.94				
		60	2.43	1.97				
I, J, P, E of Sail are define	d in Fig. 1.	80	1.39	0.98				
		100	0.15	0.10				

Table 1. Principal dimensions of Fujin and detailed measurements of sails

3. Overview of Vortex Lattice method (VLM)

3.1 Basic concept of Vortex Lattice method

The Vortex Lattice method is a branch of CFD, and it is often used at the early stage of yacht sail design because of the comparatively less computational time. This method is based on the potential theory, similar to the panel method, and the flow around the sail is expressed by discrete vortices. The Vortex Lattice method has its root in the lifting line theory formulated by Prandtl in 1918. A wing is represented by a single vortex line in the lifting line theory, and the force acting on the wing is approximated by the force acting on the vortex line.



Fig. 2. 2-Dimensional flat plate wing and a vortex filament

Firstly, 2-dimensional flow around a flat plate wing is considered. A vortex filament is located at a distance "a " from the leading edge of the wing as shown in Fig.2. Although the onset flow U is constant, the flow over the wing is accelerated, while it is decelerated below the wing, because of the flow induced by the vortex filament. This leads to the pressure decrease on the back surface and the pressure increase on the front surface of the wing accordingly to Bernoulli's theorem. This means that the flow around the wing can be realized by a vortex filament in the flow. The strength of vortex filament, or sometimes called circulation, is determined by a boundary condition on the wing; that is, there is no cross flow through the

wing. This boundary condition is usually satisfied at a certain point called control point. Assuming that the control point is located at a distance "b" from the leading edge of the wing shown in Fig.2, and satisfying the boundary condition at this point, the strength of the vortex filament can be determined. Once the strength of vortex filament is determined, the lift acting on the vortex filament can be calculated according to Kutta-Joukowski theorem, that is,

$$\mathbf{L} = \rho \mathbf{U} \boldsymbol{\Gamma} \tag{1}$$

where ρ is the density of the fluid and Γ is the strength of vortex filament. If the calculated lift is assumed to equal that generated in a 2-dimentional thin parabolic shape airfoil, the locations of the vortex filament and the control point are determined to be a = c/4 and b = 3c/4, where " c " is the chord length of the wing. This is called 1/4-3/4 rule, which was shown by Pistolesi (Pistolesi, 1937). This rule is used as the basis of the present Vortex Lattice method.

3.2 Application to sail configuration

As the yacht sail is a 3-dimension shape body, attention should be paid to the treatment of the end of vortex line. According to the Helmholtz's theorem on vortex, the vortex line should expand from the boundary to the boundary of the flow or shuts oneself and makes vortex ring. Accordingly, in the 3-dimensional body, the vortex line should expand infinity from the edge of the body. In the lifting line theory, or the Vortex Lattice method, the vortex line is assumed to be a horseshoe type shown in Fig.3(a), and the vortex line changes its direction at the edge of the body to extend to infinity as trailing free vortices.



Fig. 3. Horseshoe vortex and downwash effect

In case of the horseshoe vortex shown in Fig.3(a), the flow induced by each vortex line affects the onset flow in the magnitude and the direction. According to Biot-Savart law, the velocity vector induced by a slight part of the vortex line $d\ell$ is given by

$$\vec{\mathbf{v}} = \frac{\Gamma}{4\pi} \iint \frac{d\vec{\ell} \times \vec{\mathbf{r}}}{\left|\vec{\mathbf{r}}\right|^3} \tag{2}$$

where \vec{r} is a position vector from the vortex part to the point concerned. The downward velocity called downwash w_i is calculated by using Equation (2), which affects the onset flow. This causes the reduction of the attack angle of total inflow by α_i as shown in Fig.3(b),

and the total velocity of inflow into the wing changes to U_e . If the lift acting on the wing is defined as the force perpendicular to the onset flow direction, it is given by the following formula according to Kutta-Joukowski theorem.

$$L = L_{e} \cos \alpha_{i} \approx L_{e} = \rho U_{e} \Gamma \approx \rho U \Gamma$$
(3)

In this case, the force in the onset flow direction is generated, which is the apparent drag called induced drag given by

$$D_{i} = L_{e} \sin \alpha_{i} \approx L_{e} \alpha_{i} = \frac{L_{e}}{U} w_{i} \approx \frac{L}{U} w_{i}$$
(4)

The induced drag is a distinctive drag in a 3-dimensional wing, and does not appear in a 2-dimensional wing.

In the Vortex Lattice method, the lift, induced drag, and center of pressure are calculated by arranging horseshoe vortices of different strength on the surface of sail. By placing a number of horseshoe vortices, the sail of complex shape with twist, camber, or two or more sails, can be analyzed. Falkner used the name "Vortex Lattice" firstly in his report, in which a wing was covered with a grid of straight horseshoe vortices (Falkner, 1943, 1946). In 1950's, only the analysis where the trailing vortices are placed in the straight line was able to be carried out because of the computer capability, and the accuracy was questionable. It was 1965 when the Vortex Lattice method started to demonstrates its ability along with the development of computer, and the method came to be used for the performance prediction of yacht sail. The yacht sail is one of the most suitable objects for applying the Vortex Lattice method because of its thickness, if the viscous effect of fluid can be disregarded. An application of the Vortex Lattice method to the performance prediction of yacht sail will be explained in the following paragraph with the use of a step-by-step procedure to estimate the trailing vortex deformations.

Discretized horseshoe vortices are located on the sail plane in the Vortex Lattice method. It is usual to divide the sail plane into quadrilateral panels as shown in Fig.4, and the



Fig. 4. Panel discretization of sails

horseshoe vortices are placed at 1/4 length of panel from the front end edge of the panel so as to trail the trailing vortices rearwards. The strengths of the horseshoe vortices are determined by satisfying the boundary condition on the sail; that is, the total flow of the onset flow and the induced wake by vortices is parallel to the sail surface at control points. The control point is taken to be the point 3/4 of length of panel from the front end edge of the panel according to the 1/4-3/4 rule.

According to Biot-Savart law, the velocity vector at the control point of i-th panel induced by other vortices are given by

$$\vec{\mathbf{v}}_{i} = \frac{1}{4\pi} \int \frac{\Gamma\left(d\vec{\ell} \times \vec{\mathbf{r}}\right)}{\left|\vec{\mathbf{r}}\right|^{3}} = \sum_{j} \frac{\Gamma_{j}}{4\pi} \sum_{k=1}^{3} \frac{\cos \alpha_{kji} + \cos \beta_{kji}}{h_{kji}} \vec{\mathbf{e}}_{kji}$$
(5)

where α_{kji} , β_{kji} , h_{kji} , are the angles and the distance of a k-th filament of a j-th horseshoe vortex and i-th control point shown in Fig.5, which shows a plane containing the vortex filament and the control point. \vec{e}_{kji} is a unit vector perpendicular to the plane shown in Fig.5. k=1, 2, 3 in Equation (5) denotes each vortex filament of a horseshoe vortex. With the use of Equation (5), the boundary condition on the control point can be given by

$$\sum_{i}^{NB} \vec{v}_{i} \cdot \vec{n} = \vec{U} \cdot \vec{n}_{i} - \sum_{i}^{NW} \vec{v}_{i} \cdot \vec{n}$$
(6)

where \hat{U} is an onset flow velocity vector and \vec{n} is the unit normal vector at the control points. NB is the number of bound horseshoe vortices on the sail plane and NW is the number of trailing horseshoe vortices in the wake. Equation (6) can be written in the vector matrix form by

$$[\Lambda]{\Gamma} = {u_n} + {v_n}$$
(7)

Solving Equation (7), the strength of bound vortices can be obtained.



Fig. 5. K-th filament of j-th horseshoe vortex and i-th control point

3.3 Step-by-step vortex shedding technique

The important point in the Vortex Lattice approach to yacht sail is the handling of the wake of the sail. The wake vortices proceed downstream from the trailing edge, or leech/foot of sail, in the Vortex Lattice method. The location of wake vortices are determined by the condition that they are free vortices; that is, the stream line of wake vortices should be parallel to the velocity field induced by total vortex system. A step-by-step procedure developed by Fukasawa was adopted in this paper to determine the strength of the bound vortices and the location of wake vortices (Fukasawa, 1993; Fukasawa & Katori, 1993; Masuyama et al., 1997a, 1997b). The wake vortices are shed from the trailing edge in each time step according to Helmholtz's theorem; that is,

$$\Gamma_{\rm B} + \Gamma_{\rm W} = 0 \tag{8}$$

$$\frac{D\Gamma_{B}}{Dt} = 0 \tag{9}$$

where Γ_B and Γ_W are the total strength of bound vortices, or the circulation around the sail, and the strength of wake vortices, respectively. From Equation (9), we have

$$\frac{\partial \Gamma_{\rm B}}{\partial t} = \tilde{U} \frac{\partial \Gamma_{\rm B}}{\partial \xi} \tag{10}$$

and substituting Equation (10) into (8), we have

$$\frac{\partial \Gamma_{\rm B}}{\partial t} = -\tilde{U} \frac{\partial \Gamma_{\rm W}}{\partial \xi} \tag{11}$$

where ξ is taken to the downstream direction, and \tilde{U} is the local velocity at the wave vortex. Assuming that the wake vortices proceeds $\Delta \xi$ downstream in a time step Δt with the velocity \tilde{U} , the strength of wake vortex shed at time step k, can be given by integrating Equation (11), that is,

$$\Delta \Gamma_{W}^{k} = -\frac{1}{\tilde{U}} \int_{0}^{\Delta t} \frac{\partial \Gamma_{B}}{\partial t} d\xi = -\frac{\Delta \xi}{\tilde{U} \Delta t} \left(\Gamma_{B}^{k} - \Gamma_{B}^{k-1} \right) = \Gamma_{B}^{k-1} - \Gamma_{B}^{k}$$
(12)

Equation (12) means that the strength of wake vortex shed at time step k is the increase of the strength of bound vortex from time step k-1 to time step k. Once a vortex filament is shed at time step k, it proceeds downstream with a constant strength according to the local field velocity, i.e., each horseshoe wake vortex moves in the direction of field velocity in each time step. The field velocity is updated in every time step. The calculation is carried forward until the the calculated lift and drag forces converges. The forces vector and the moment acting on the sail are calculated accordingly to Kutta-Joukowski theorem.

$$\vec{F} = \rho \int \Gamma \tilde{U} \times d\vec{s}$$
(13)

with the use of the vortex strengths of the wake vortices and the bound vortices determined by solving Equation (7).

Finally, the overall numerical solution procedure of the present Vortex Lattice method is summarized as follows:

- Step 1. Divide the sail planes into quadrilateral panels, and allocate horseshoe vortices on the sail plane.
- Step 2. Input the mast rake angle, heel angle of the yacht, apparent wind speed and apparent wind angle.

- Step 3. Solve the strength of bound horseshoe vortices on the sail plane with nondeformed trailing vortices.
- Step 4. Compute the total circulation around the sail caused by horseshoe vortices.
- Step 5. Compute the increment of the total circulation and shed free horseshoe vortices according to Equation (12).
- Step 6. Calculate the local velocities along the wake, and deform the trailing vortices.
- Step 7. Compute the force vector and moment acting on the sails.
- Step 8. Solve the strength of bound horseshoe vortices on the sail plane with trailing free vortices.
- Step 9. Repeat Step 4 through Step 8 until the force is converged.

Fig.6 shows the example calculation results. In the present study, the mast and rigging were not considered for the series calculations, and the mirror image was taken into account about the deck plane of the boat. Since the vortex lattice methods do not predict viscous drag, the viscous drag acting on the sails and rigging was calculated empirically using a drag coefficient C_{Dp} . The value of C_{Dp} was obtained from the measured data in the previous papers and formulated for the upwind condition as follows:

$$C_{\rm Dp} = 0.0026 \,\gamma_{\rm A} + 0.005 \tag{14}$$

where γ_A is apparent wind angle in degrees.



Fig. 6. Calculated wake by Vortex Lattice method

4. Overview of Reynolds-averaged Navier-Stokes equation method

The RANS-based CFD method used in the present study was *FLOWPACK*. The code was developed by Tahara specifically for CFD education and research and for design applications for ship hydrodynamics, aerodynamics, and fluid engineering. As part of the developments for application to design problems, a complete multiblock domain

decomposition feature was included. At present, *FLOWPACK* has a good interface with the authors' inhouse automatic grid generator as well as with commercial grid generation software. For a complete documentation of the method is available in Tahara (2008). In the following, an overview of the numerical method is given.

4.1 Governing equations

Let us consider a sail system fixed in the uniform onset flow (see Fig.1 for the basic coordinate system). The non-dimensional RANS equations for unsteady, three-dimensional incompressible flow can be written in Cartesian tensor notation as

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x^j} + \frac{\partial u_i u_j}{\partial x^j} + \frac{\partial p}{\partial x^i} - \frac{1}{\text{Re}} \nabla^2 U_i = 0$$
(15)

$$\frac{\partial U_i}{\partial x^i} = 0 \tag{16}$$

where U_i (*i*=1,2,3) =(*U*,*V*,*W*) and u_i (*i*=1,2,3) =(*u*,*v*,*w*) are the Cartesian components of mean and fluctuating velocities, respectively, normalized by the reference velocity U_0 , x^i (*i*=1,2,3) =(*X*,*Y*,*Z*) is the dimensionless coordinates normalized by a characteristic length *L*, Re= U_0L/v is the Reynolds number, *v* is the kinematic viscosity, the barred quantities $-u_iu_j$ are the Reynolds stresses normalized by U_0^2 , and *p* is the pressure normalized by ρU_0^2 . If $-u_iu_j$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity v_t , i.e.,

$$-\overline{u_i u_j} = v_t \left(\frac{\partial U_i}{\partial x^j} + \frac{\partial U_j}{\partial x^i} \right) - \frac{2}{3} \delta_{ij} k$$
(17)

where k = (uu + vv + ww)/2 is the turbulent kinetic energy, Equation (15) becomes

$$\frac{\partial U_i}{\partial t} + \left(U_j - \frac{\partial v_t}{\partial x^j}\right) \frac{\partial U_i}{\partial x^j} - \frac{\partial v_t}{\partial x^j} \frac{\partial U_j}{\partial x^i} + \frac{\partial p}{\partial x^i} \left(p + \frac{2}{3}k\right) - \frac{1}{R_{\phi}} \nabla^2 U_i = 0$$
(18)

where $1/R_{\phi} = 1/\text{Re} + v_t$, and $\phi = U_i$ (*i*=1,2,3). Equations (16) and (18) can be solved for U_i and p when a suitable turbulence model is employed to calculate the eddy-viscosity distribution. Either a zero or a two-equation turbulence model can be used for turbulent flow calculation, and a model used for the present study is the former, i.e., Baldwin-Lomax model (Baldwin & Lomax, 1978), which is an algebraic scheme that makes use of a two-layer isotropic eddy-viscosity formulation. Detailed validation study of this model for boundary layer flows around three-dimensional bodies was done by the author (Tahara, 1995; Tahara & Stern, 1996). In this model, the eddy viscosity is evaluated as follows:

$$\begin{cases} (v_t)_{inner} & y \le y_c \\ (v_t)_{outer} & y > y_c \end{cases}$$
(19)

where y is the distance normal to the wall surface and y_c is the minimum value of y where both the inner and outer viscosities match. The inner viscosity follows the Prandtl-Van

Driest formula, i.e., $(v_t)_{inner} = \lambda^2 |\omega|$, where $\lambda = \kappa y [1 - \exp(-y^+/A^+)]$ is the turbulent length scale for the inner region, κ and A^+ are model constants, $|\omega|$ is the vorticity magnitude, and y^+ is the dimensionless distance to the wall. In the outer region, eddy viscosity is given by $(v_t)_{outer} = KC_{cp}F_{wake}F_{Kleb}$, where K and C_{cp} are model constants, $F_{wake} = \min(y_{max}F_{max}, C_{wk}y_{max}U_{dif}^2/F_{max})$, and $F_{Kleb} = [1+5.5(C_{Kleb}y/y_{max})^6]^{-1}$. The F_{max} and y_{max} are determined by the value and corresponding location, respectively, of the maximum of $F=y |\omega| [1-\exp(-y^+/A^+)]$. The quantity U_{dif} is the difference between maximum and minimum velocity magnitudes in the profile and is expressed as $U_{dif} = (U^2 + V^2 + W^2)_{max}^{1/2} - (U^2 + V^2 + W^2)_{min}^{1/2}$. C_{Kleb} and C_{wk} are additional model constants. Numerical values for the model constants are $A^+=26$, $\kappa=0.4$, K=0.0168, $C_{cp}=1.6$, $C_{wk}=1.0$, and $C_{Kleb}=0.3$.

4.2 Discretization and velocity-pressure coupling

In the following, discretization and velocity-pressure coupling of the present RANS method are described. First, it is convenient to rewrite the transport equations for momentum (U_i) in the following general form:

$$\nabla^2 \phi = R_{\phi} \left[\sum_{j=1}^3 \left(U_j - \frac{1}{\sigma_{\phi}} \frac{\partial v_t}{\partial x^j} \right) \frac{\partial \phi}{\partial x^j} + \frac{\partial \phi}{\partial t} \right] + s_{\phi}$$
(20)

where ϕ again represents any one of the convective transport quantities (U_i), and s_{ϕ} is the source function for the corresponding quantity. We transform the physical space (x^i, t) into a rectangular region in the computational space (ξ^i, τ) using the following coordinate transformations:

$$t = \tau, \quad x^{i} = x^{i} \left(\xi^{j}\right), \qquad \qquad e_{i} \cdot \nabla \phi = \frac{1}{J} \sum_{j=1}^{3} b_{i}^{j} \frac{\partial \phi}{\partial \xi^{j}}$$

$$\nabla^{2} \phi = \sum_{i=1}^{3} \sum_{j=1}^{3} g^{ij} \frac{\partial^{2} \phi}{\partial \xi^{i} \xi^{j}} + \sum_{j=1}^{3} f^{j} \frac{\partial \phi}{\partial \xi^{j}}, \qquad \frac{\partial \phi}{\partial t} = \frac{\partial \phi}{\partial \tau} - \frac{1}{J} \sum_{i=1}^{3} \sum_{j=1}^{3} b_{i}^{j} \frac{\partial x^{i}}{\partial \tau} \frac{\partial \phi}{\partial \xi^{j}}$$

Then the continuity equation (16) and the transport equations (20) for momentum parameters can be written as

$$\frac{1}{J}\sum_{i=1}^{3}\sum_{j=1}^{3}\frac{\partial}{\partial\xi^{i}}\left(b_{j}^{i}U_{j}\right)=0$$
(21)

$$\sum_{j=1}^{3} \left(g^{jj} \frac{\partial^2 \phi}{\partial \xi^j \partial \xi^j} - 2a_{\phi}^j \frac{\partial \phi}{\partial \xi^j} \right) = R_{\phi} \frac{\partial \phi}{\partial \tau} + S_{\phi}$$
(22)

where

$$2a_{\phi}^{j} = \frac{R_{\phi}}{J} \sum_{n=1}^{3} b_{n}^{j} \left(U_{n} - \frac{\partial x^{n}}{\partial \tau} - \frac{1}{J\sigma_{\phi}} \sum_{m=1}^{3} b_{n}^{m} \frac{\partial v_{t}}{\partial \xi^{m}} \right) - f^{j}$$

100

$$S_{\phi} = s_{\phi} - 2 \left(g^{12} \frac{\partial^2 \phi}{\partial \xi^1 \partial \xi^2} + g^{13} \frac{\partial^2 \phi}{\partial \xi^1 \partial \xi^3} + g^{23} \frac{\partial^2 \phi}{\partial \xi^2 \partial \xi^3} \right)$$

The geometric coefficients b_i^j , g^{ij} , and f^j appearing in the above equations are defined by Thompson et al. (1985). The transport equations (22) on a computational cell (shown in Fig.7(a)) are linearized and evaluating coefficients and source term at the center node P of the element yields

$$\sum_{j=1}^{3} \left(g_{\mathrm{P}}^{jj} \frac{\partial^{2} \phi}{\partial \xi^{j} \partial \xi^{j}} - 2 \left(a_{\phi}^{j} \right)_{\mathrm{P}} \frac{\partial \phi}{\partial \xi^{j}} \right) = \left(R_{\phi} \right)_{\mathrm{P}} \frac{\partial \phi}{\partial \tau} + \left(S_{\phi} \right)_{\mathrm{P}}$$
(23)

or

$$g_{\rm P}^{11}\phi_{\xi^{1}\xi^{1}} + g_{\rm P}^{22}\phi_{\xi^{2}\xi^{2}} + g_{\rm P}^{33}\phi_{\xi^{3}\xi^{3}} = 2(C_{\phi})_{\rm P}\phi_{\xi^{1}} + 2(B_{\phi})_{\rm P}\phi_{\xi^{2}} + 2(A_{\phi})_{\rm P}\phi_{\xi^{3}} + (R_{\phi})_{\rm P}\phi_{t} + (S_{\phi})_{\rm P}$$
(24)



Fig. 7. Definition sketch of a computational cell (a), and nodes in regular grid (b) and continuity cell (c).

The dimensions of the computational cell are $2l \times 2k \times 2h$, where $l=1/\sqrt{g_P^{11}}$, $k=1/\sqrt{g_P^{22}}$, and $h=1/\sqrt{g_P^{33}}$. The above equation is discretized by the finite-analytic scheme. Solution dependent coefficients are analytically derived by solving the above linearized transport equation using a hybrid method which combines a two-dimensional analytic solution in $\xi^2 \xi^3$ -plane with one dimensional analytic solution in the ξ^1 direction. By specifying boundary conditions on the faces of the cell as a combination of exponential and linear functions, which are the natural solutions for the linearized transport equation, Equation (24) can be solved by the method of separation of variables. When the solution is evaluated at the center node P of the element, the following twelve-point finite analytic formula is obtained:

$$\phi_{P}^{n} = \frac{1}{1 + C_{P} \left(C_{U} + C_{D} + \frac{R\phi}{\Delta\tau} \right)} \left| \begin{array}{c} C_{NE} \phi_{NE}^{n} + C_{NW} \phi_{NW}^{n} + C_{SE} \phi_{SE}^{n} + C_{SW} \phi_{SW}^{n} \\ + C_{EC} \phi_{EC}^{n} + C_{WC} \phi_{WC}^{n} + C_{NC} \phi_{NC}^{n} + C_{SC} \phi_{SC}^{n} \\ + C_{P} \left(C_{U} \phi_{U}^{n} + C_{D} \phi_{D}^{n} + \frac{R_{\phi}}{\Delta\tau} \phi_{P}^{n-1} \right) - C_{P} \left(S_{\phi} \right)_{P} \right]$$
(25)

where

$$C_{\rm U} = \frac{Ce^{Cl}}{l\sinh Cl}, \ C_{\rm D} = \frac{Ce^{-Cl}}{l\sinh Cl}, \ C_{\rm SC} = \frac{e^{Bk}}{2\cosh Bk} P_A, \ C_{\rm NC} = e^{-2Bk}C_{\rm SC}$$

$$C_{\rm WC} = \frac{e^{Ah}}{2\cosh Ah} P_B, \ C_{\rm EC} = e^{-2Ah}C_{\rm WC}, \ C_{\rm SW} = \frac{e^{Ah+Bk}}{4\cosh Ah\cosh Bk} (1 - P_A - P_B)$$

$$C_{\rm SE} = e^{-2Ah}C_{\rm SW}, \ C_{\rm NW} = e^{-2Bk}C_{\rm SW}, \ C_{\rm NE} = e^{-2Ah-2Bk}C_{\rm SW}$$

$$C_{\rm P} = \frac{h\tanh Ah}{2A} (1 - P_A) = \frac{k\tanh Bk}{2B} (1 - P_B), \ P_A = 4E_2Ah\cosh Ah\cosh Bk \coth Ah$$

$$P_{B} = 1 + \frac{Bh \coth Bk}{Ak \coth Ah} (P_{A} - 1) , \quad E_{2} = \sum_{m=1}^{\infty} \frac{-(-1)^{m} (\lambda_{m}h)}{\left[\left(Ah\right)^{2} + \left(\lambda_{m}h\right)^{2}\right]^{2} \cos\sqrt{A^{2} + B^{2} + \lambda_{m}^{-2}k}}$$
$$\lambda_{m}h = \left(m - \frac{1}{2}\right)\pi$$

The subscripts P, U and D denote the center, upstream and downstream nodes, respectively, and NC, NW, WC, etc. denote the nodes in the $\xi^2 \xi^3$ -plane in terms of compass directions. The superscripts (n) and (n-1) refer to the current and previous time levels, and $\Delta \tau$ is the time step. The solution of the complete flow equations involves a global iteration process, in which the velocity-pressure coupling is effected by PISO-type predictor-corrector steps. The pressure equation is derived by introducing pseudo-velocities at staggered locations while maintaining the regular grid arrangement for all the transport equations. Fig.7(b) and (c) show the locations of nodes in the regular grid in the $\xi^2\xi^3$ -plane. All transport quantities and pressure are evaluated at the regular nodes. In deriving the pressure equation, a control volume is employed as a continuity cell, to establish the coupling between the velocity and pressure fields. The pressure equation used in this study is written as

$$\begin{pmatrix} E_{\rm d}^{11} + E_{\rm u}^{11} + E_{\rm n}^{22} + E_{\rm s}^{22} + E_{\rm e}^{33} + E_{\rm w}^{33} \end{pmatrix} p_{\rm P} = E_{\rm d}^{11} p_{\rm D} + E_{\rm u}^{11} p_{\rm U} + E_{\rm n}^{22} p_{\rm NC} + E_{\rm s}^{22} p_{\rm SC} + E_{\rm e}^{33} p_{\rm EC} + E_{\rm w}^{33} p_{\rm WC} - \hat{D}$$

$$(26)$$

with

$$\hat{D} = \hat{U}_{\rm d}^1 - \hat{U}_{\rm u}^1 + \hat{U}_{\rm n}^2 - \hat{U}_{\rm s}^2 + \hat{U}_{\rm e}^3 - \hat{U}_{\rm w}^3$$

Here E^{ij} and a modified pseudovelocity \hat{U}^i at the regular node are

$$E^{ij} = \frac{R C_{\rm P}}{J \left[1 + C_{\rm P} \left(C_{\rm U} + C_{\rm D} + \frac{R}{\Delta \tau} \right) \right]} \sum_{m=1}^{3} b^i_m b^j_m , \ \hat{U}^i = \sum_{n=1}^{3} b^i_n \hat{U}_n - E^{ij} \frac{\partial p}{\partial \xi^j} - E^{ik} \frac{\partial p}{\partial \xi^k}$$
(27)

where \hat{U}_i is a pseudovelocity given by the decomposition of Equation (25) for U_i into \hat{U}_i plus the pressure gradient terms, such that

$$U_{i} = \hat{U}_{i} - \frac{R C_{P}}{J \left[1 + C_{P} \left(C_{U} + C_{D} + \frac{R}{\Delta \tau} \right) \right]} \sum_{j=1}^{3} b_{i}^{j} \frac{\partial p}{\partial \xi^{j}}$$
(28)

The coefficients and the modified pseudovelocities in the above equations are defined at the staggered node, and obtained from those at the regular node by the one-dimensional linear interpolation. The solution of the complete flow equations involves a global iteration process, in which the velocity-pressure coupling is effected by predictor-corrector steps. In the predictor step, the pressure field at the previous time step is used in the solution of the implicit equations (25) to obtain the corresponding velocity field. Since the velocity field generally does not satisfy mass conservation, a corrector step is needed. In the corrector step, the explicit momentum equations (28) and the implicit pressure equation (26) are solved iteratively to ensure the satisfaction of the continuity equation.

4.3 Multiblock (domain decomposition) capability, and overall numerical solution procedure

As mentioned earlier, the multiblock (domain decomposition) capability is facilitated in the present RANS method. This capability is essential for simulation of flow around complex geometry, e.g., multiple sail system for sailing yacht as focused in the present study. Fig.8 shows overview of the present multiblock computational grid, while the grid is generated by using an automatic gridding scheme developed by the present author (Masuyama et al., 2009) Note that the gridding engine together with the present RANS method was recently implemented into a comprehensive sail performance prediction software "*Advanced Aero Flow*" (Katori, 2009). See the reference for more details of the scheme. Total number of grids is around a half million, and the number of multiblock is 48. Free-stream, symmetry, and wall-surface (no slip) boundary conditions are imposed on outer and top boundaries, bottom boundary, and sail surface boundary, respectively. For the results shown in this paper, the mast and rigging are not considered in the series calculations, and the bottom boundary is located at the same height as that of deck plane of the boat (see Masuyama et al., 2009, for the results for which mast influences in computation are considered).



Fig. 8. Overview of the present multiblock computational grid.

The basic strategy to handle the multiblock follows domain decomposition technique to solve the elliptic PDE by using several subdomains. After adequate discretization is applied and a simple preconditioner is introduced, the discrete alternating Schwarz's method to solve the PDE is used for boundary matching. Finally, Fig.9 shows the code structure of the present RANS method, and the overall numerical solution procedure of the present RANS method is summarized as follows:



Fig. 9. PISO type solution algorithm for the present multiblock RANS method.

- Step 1. Input the computational grid, setup parameters, and boundary condition information.
- Step 2. Specify the initial conditions for the velocity, pressure and turbulence fields.
- Step 3. Compute the geometric coefficients.
- Step 4. Compute the finite-analytic coefficients for the transport equation.
- Step 5. Compute eddy viscosity distribution.
- Step 6. Solve transport equation for velocities (U, V, W) using the previous pressure field (predictor stage for the velocity field).
- Step 7. Compute the coefficients of pressure equation.
- Step 8. Solve pressure equation.
- Step 9. Using the newly obtained pressure, calculate the new velocity field explicitly (corrector stage for the velocity field).
- Step 10. Update the finite-analytic coefficients for the transport equation for velocities (U, V, W).

Step 11. Repeat Step 8 through Step 10 for the specified number of times.

Step 12. Return to Step 4 for the next time step, until the time step reaches the given maximum value.

More details of the present RANS method are described in Tahara et al. (2006a, 2006b) in addition to the above-cited references.

5. Measurements of upwind sail performance in full-scale condition using sail dynamometer boat *Fujin*

5.1 Full-scale measurements

Full-scale onboard measurements are free from scale-effect problem by wind tunnel tests and appear more promising, but the challenge becomes how to accurately measure forces acting on the sail. Such studies on sail force measurements were performed by Milgram et al. (1993), Masuyama et al. (1997a, 1997b), and Hochkirch et al. (1999), who built full-scale boats with onboard sail dynamometer systems.

Milgram (1993) showed in his pioneering work that the sail dynamometer boat, *Amphetrete*, is quite capable. This measurement system consists of a 35-foot boat with an internal frame connected to the hull by six load cells, which were configured to measure all forces and moments acting on the sails. In his work, the sail shapes were also measured and used for CFD analyses; however unfortunately, details of the sail shape and performance data were not presented. Hochkirch et al. (1999) also built a 33-foot dynamometer boat *DYNA*. The aerodynamic forces acting on the sail were measured and compared with the results from wind tunnel tests (Hansen et al. 2003). The measured data were also used as input to the CFD calculation and a parametric survey was carried out (Krebber et al. 2006). Masuyama and Fukasawa were encouraged by Milgram's work, and built a sail dynamometer boat, *Fujin*. The measurement system installed in the *Fujin* and the results of calibration test and sailing test were reported by Masuyama et al. (1997a and 1997b).

5.2 Measurements by sail dynamometer boat Fujin

The *Fujin* was originally built for conducting tests on sails for the Japanese America's Cup entry in 1994. *Fujin* is a 10.3m-long ocean cruiser with a sail dynamometer system in the hull which can directly measure sail forces and moments. Fig. 1 shows the general arrangement of the *Fujin*. The test sails were made to correspond to a typical sail plan for an International Measurement System (IMS) class boat. The rigging of the *Fujin* was originally designed for testing sails for the International America's Cup Class (IACC) boat. The jib of IACC boat is relatively small. Therefore, the longitudinal position of the jib rail track of the *Fujin* was located further forward than that of the typical IMS boat. For this reason, the tests were performed using a fully batten mainsail and a 130% jib instead of a 150% jib. The sails were made by North Sails Japan. The axes system is also shown in Fig. 1. The origin is located on the vessel's centerline at the aft face of the mast (*x*-direction), and the height of deck level at the base of the forestay (*z*-direction). Table 1 shows the principal dimensions of the boat and the detailed measurements of the sails, where "I", "J", "P" and "E" are the measurement lengths of sail dimensions for the IMS rule as defined in Fig.1.

The aerodynamic coefficients and the coordinates of the center of effort of the sails are defined as follows:

$$C_{X} = \frac{X_{S}}{\frac{1}{2}\rho_{a}U_{A}^{2}S_{A}}, \qquad C_{Y} = \frac{Y_{S}}{\frac{1}{2}\rho_{a}U_{A}^{2}S_{A}}, \qquad x_{CE} = \frac{N_{S}}{Y_{S}}, \qquad z_{CE} = \frac{K_{S}}{Y_{S}}$$
(29)

where X_S and Y_S are the force components along the *x* and *y* axes of the boat respectively, and K_S and N_S are the moments around the *x* and *z* axes. x_{CE} and z_{CE} are the *x* and *z* coordinates of the center of effort of the sails (CE). The thrust force coefficient C_X is expressed as positive for the forward direction and the side force coefficient C_Y is positive for both port and starboard directions. It should be noted that the coordinates are given in the body axes system. Therefore, when the boat heels the Y_S force component is not in the horizontal plane but is normal to the mast. The aerodynamic forces acting on the mast and rigging are included in the measured sail forces.

5.2.1 Measurement system of aerodynamic performance and sail shape

The sail dynamometer system is composed of a rigid aluminum frame and four load cells. The frame is separated structurally from the hull and connected to it by the load cells. The general arrangement of the dynamometer frame is given in Fig.10. The load cells are numbered in the figure. Two of these are 1-component load cells and the others are 2-component ones. Hence, these load cells form a 6-component dynamometer system, and their outputs can be transformed to the forces and moments about the boat axes using a calibration matrix. All rig components such as mast, chain plates, winches, lead blocks, etc. are attached to the aluminum frame. The under deck portion of the mast is held by the frame, and the other rig components are attached to the frame through the deck holes. The data acquisition system and calibration method for the *Fujin* were described by Masuyama et al. (1997a and 1997b).

The sail shape was recorded using pairs of CCD cameras. The lower part of the mainsail was photographed using the CCD camera pair designated A in Fig.11. These were located at the mast top, 50 cm transversely from each side of the mast. The upper part of the mainsail was photographed using a portable video camera from below the boom. The lower part of the jib was photographed using the camera pair designated B in Fig.11, which were located at the intersection point of the forestay and the mast, 10 cm transversely from each side of the mast. The upper part of the jib was photographed using a portable video camera from inside the bow hatch. For measuring convenience, horizontal stripes were drawn on the mainsail and jib at heights of 10, 20, 40, 60 and 80% of each sail. The sail shape images were analyzed using the sail shape analyzing software, SSA-2D, developed by Armonicos Co. Fig.12(a) shows an example of processed image of the mainsail using the SSA-2D. This software calculates the curvature of the sail section by marking several points of the sail stripe and the reference line on the PC display, and indicates the parameters such as chord length, maximum draft, maximum draft position, entry angle at the luff, i.e., leading edge, and exit angle at the leech, i.e., trailing edge, as shown in Fig.12(b). The apparent wind speed (AWS) and apparent wind angle (AWA) are measured by an anemometer attached on the "Bow unit" as shown in Fig.11. This unit post can rotate freely to maintain its vertical attitude when the boat heels in order to measure the wind data in the horizontal plane. The height of the anemometer coincides with the geometric center of effort (GCE) of the sail plan. The wind speed and wind angle sensors were calibrated by wind tunnel tests in advance and the calibration equations were obtained. The Fujin also has motion measuring instruments such

as an Optical Fiber Gyroscope (roll and pitch angles), a Flux Gate Compass (heading angle), a Differential type GPS receiver, a speedometer (velocity in the x direction) and a potentiometer for rudder angle. These data are recorded by an onboard computer simultaneously with the data from the load cells.



Fig. 10. General arrangement of dynamometer frame in Fujin.



Fig. 11. Sea trial condition in light wind with 130% jib.



Fig. 12. (a) Example of a processed image of the mainsail using SSA-2D. (b) Measured sail shape parameters.

5.2.2 Test condition and error analysis

The sea tests were performed on Nanao Bay off the Noto Peninsula. The bay is approximately eight nautical miles from east to west and five from north to south. The bay is surrounded by low hills, and the mouth connecting it to the Japan Sea is narrow. Therefore, there is little tidal current in the bay, and the wave heights are relatively low even though the wind can be strong. The close-hauled tests were conducted over the apparent wind angle (AWA) range of 20 to 40 degrees, and the apparent wind speed (AWS) range of 5 to 11m/s. The effect of the AWA, and the draft and twist of the mainsail on the sail performance were measured. Data sampling was started when the sailing condition was considered to be in steady state. The sampling rate for the data acquisition system was set at 10Hz. Data sampling was continued for 90 seconds, and during this time the sail shapes were recorded using the CCD cameras. The boat was steered carefully during this time. However, the measured data contained some variation due to wind fluctuation and wave reflection on the hull. Therefore the steady state values for the aerodynamic coefficients were obtained by averaging the data over a 30 to 60 seconds period, in which the AWA was closer to the target value than during the whole 90 second period. For these tests if the range of deviation of AWA exceeded ±5 degrees, the results were discarded. All of the measured coefficients are plotted with error bars indicating the range of deviation over the averaging period.

5.3 Comparison between experimental and calculated results

In this chapter, the experimental results and the calculated results for the following cases will be compared:

- a. Variation with apparent wind angle
- b. Variation with mainsail twist angle

For each series, first the sail coefficients: C_L , C_D , C_X and C_Y , and the coordinates of x_{CE} and z_{CE} are given. Then, the calculated the sail surface pressure and streamlines using the RANSbased CFD are presented for two typical cases in each series. Finally, the shapes and threedimensional coordinates of the sails are tabulated for each case corresponding to those where the RANS-based CFD results are given.

5.3.1 Variation with apparent wind angle

Fig.13 shows the performance variation for the mainsail and 130% jib configuration as a function of AWA. In the figure the solid symbols indicate the experimental results and the open symbols indicate the calculated results using the VLM and the RANS-based CFD. For the experimental results, both data from the starboard (Stbd) and port tack (Port) are shown. All of the measured coefficients are plotted with error bars indicating the range of deviation over the averaging period. There are some discrepancies between the data from each tack. During the experiments, efforts were made to remove this asymmetrical performance. However, the boat speed actually differed on each tack. It can be concluded that there was a slight asymmetry in the combination of the hull, keel, rudder and dynamometer frame. The numerical calculations were performed using the measured shape data. In order to avoid confusion when interpreting the figure, the calculated results are indicated only for the port tack. Therefore, the calculated and experimental points for the port tack correspond to each other.



Fig. 13. Performance variation as a function of apparent wind angle (AWA) for mainsail and 130% jib.

In this figure, AWA ranges from 20.3 degrees to 37.9 degrees for the port tack. The former is the closest angle to the wind that was achieved, and the latter is typical of a close reaching condition, where the sail is eased for the power down mode. There is some scatter in the experimental data because this is made up from measurements taken with the sails trimmed in slightly different ways. The experimental value of C_L in Fig.13(a) varies with AWA from 0.91 to 1.58. For the close reaching condition, the sails were not well trimmed to satisfy the power down mode. A sample of measured sail sections at this condition is shown in a figure attached to Table 2(2). From the figure, it can be seen that both the mainsail and the jib are not eased sufficiently to correspond to the large AWA. This is the reason for the decrement in the measured lift curve slope of C_L at the range of AWA angles over about 35 degrees.

1	(1) 96092335				(Z)	WIST det	48 DRAFT[3]	AWS[m/s]HEEL deal VB [kt]					
	30.7	15.5	8.6	6.9	15.1	5.0		37.9	14.5	7.2	7.5	19.6	6.0
	C,	Gn	C.	C,	Xcr [m]	ZOF [m]		C,	Co	C,	C _v	Xer [m]	Zes [m]
	1.44	0.28	0.50	1.39	0.41	4.17		1.58	0.45	0.62	1.52	0.34	4.1
6 of		130%Jib		1	Mainsail		5 of	1	30%Jib			Mainsail	
heit	×	y	2	x	v	2	heit	x	V	2	*	V	Z
	-3.780	0.000	0.000	0.046	0.000	1.320		-3.780	0.000	0.000	0.046	0.000	1.320
1.1	-2.812	0 136	0.000	0.934	0.000	1,320		-2.812	0.136	0.000	0.934	0.015	1.320
Ð.	-1.843	0.272	0.000	1.822	0.000	1.320	0	-1.843	0.272	0.000	1.822	0.031	1.320
N.	-0.875	0.408	0.000	2,710	0.000	1.320		-0.875	0.408	0.000	2,710	0.046	1.320
(9)	0.094	0.544	0.000	3 598	0.000	1.320		0.094	0.544	0.000	3.597	0.062	1.320
	1.062	0.681	0.000	4 486	0.000	1 320		1.062	0.681	0.000	4.485	0.077	1.320
	-2.998	0.000	2,140	0.133	0.000	3.820	-	-2.998	0.000	2 140	0.133	0.000	3.820
11	-2 305	0.429	2,140	0 888	0.176	3.820		-2314	0.461	2140	0.891	0.150	3.820
20	-1 568	0.667	2.140	1.645	0.322	3.820	20	-1.597	0.750	2.140	1.651	0 267	3 820
5	-0.805	0 795	2 140	2 406	0.400	3.820	4	-0.841	0.840	2 140	2 414	0.331	3 820
1	-0.027	0.861	2 140	3.173	0.363	3.820	1.00%	-0.062	0.810	2.140	3.182	0.333	3.820
	0.760	0.886	2.140	3.947	0.222	3,820	-	0.728	0.724	2.140	3.954	0.262	3 820
12	-2.215	0.000	4,280	0.221	0.000	6.320	1.00	-2 215	0.000	4,280	0.221	0.000	6.320
	-1.771	0.442	4,280	0.834	0.227	6.320	40	-1.769	0.437	4,280	0.829	0.239	6.320
40	-1.272	0.719	4,280	1.452	0.405	6.320		-1.274	0.729	4,280	1.445	0.423	6.320
X.	-0.723	0.850	4,280	2.081	0.483	6.320	N.	-0.726	0.863	4.280	2.074	0.520	6.320
	-0.145	0.898	4,280	2722	0.442	6.320		-0.145	0.899	4.280	2,717	0.511	6.320
	0.448	0.898	4,280	3.371	0.331	6.320		0.450	0.892	4.280	3,368	0.442	6.320
	-1.433	0.000	6,420	0.308	0.000	8.820		-1.433	0.000	8.420	0.308	0.000	8.820
1.5	-1.186	0.332	6.420	0.761	0.218	8.820	1.1	-1,218	0.362	6.420	0.757	0.230	8.820
60	-0.893	0.570	6.420	1.222	0.389	8.820	60	-0.940	0.615	6.420	1.218	0.397	8.820
\$	-0.552	0.715	6,420	1.699	0.470	8.820	5	-0.601	0.763	6.420	1.697	0.482	8.820
	-0.176	0.790	6.420	2.191	0.462	8.820		-0.230	0.854	6.420	2.187	0.504	8.820
	0.217	0.832	6.420	2.691	0.410	8,820		0.157	0.918	6.420	2.687	0.481	8.820
	-0.650	0.000	8.560	0.396	0.000	11.320	1.3	-0.650	0.000	8,560	0.396	0.000	11.320
	-0.541	0.172	8.560	0.651	0.144	11.320	80	-0.565	0.191	8,560	0.656	0.128	11.320
80	-0.414	0.318	8,560	0.914	0.261	11.320		-0.445	0.339	8,560	0.921	0.241	11.320
*	-0.255	0.419	8.560	1.190	0.330	11.320		-0.289	0.444	8.560	1.193	0.327	11.320
	-0.073	0.486	B.560	1.476	0.362	11.320		-0.113	0.527	8 560	1.478	0.368	11.320
	0.122	0.535	8.560	1.768	0.374	11.320		0.071	0.597	8,560	1.771	0.377	11.320
	0.132	0.000	10,700	0.483	0.000	13.820		0.132	0.000	10,700	0.483	0.000	13.820
	0.144	0.016	10.700	0.511	0.012	13.820		0,142	0.018	10,700	0.511	0.011	13.820
100	0,159	0.030	10,700	0.538	0.023	13.820	100	0.154	0.034	10,700	0.539	0.022	13.820
5	0.173	0.044	10,700	0.567	0.033	13.820	5	0.167	0.049	10,700	0.567	0.032	13.820
2	0.189	0.056	10,700	0.595	0.042	13.820		0.181	0.064	10,700	0.596	0.041	13.820
121	0.207	0.066	10,700	0.624	0.051	13.820	-	0.196	0.077	10,700	0.625	0.049	13.820

Table 2. Sail shapes, measured experimental data and three-dimensional coordinates of the sails for the cases of (1) 96092335 and (2) 96080248.

The calculated results for C_L using the VLM show good agreement with the experiments at AWA angles less than about 35 degrees. Over about 35 degrees, the calculated results are lower than the measured ones. This shows that the calculated results strongly indicate the effect of incorrect sail trimming. The results for C_L using the RANS-based CFD show the same trends with the experiments, but are slight higher than those from the experiments for AWA between 20 degrees to 30 degrees and lower for AWA greater than 30 degrees. In particular, the decrease in C_L for AWA values greater than 30 degrees is considerably large. This will be discussed later with the calculated sail surface pressure and streamlines. The calculated results for C_D slightly over predict those from the experiments. Fig.13(c) shows the coordinates of the center of effort of the sails. The *x* and *z* coordinates of the geometric center of effort (x_{GCE} and z_{GCE}) are 0.63m aft and 4.80m above the origin, which are indicated by alternate long and short dashed lines in the figure. It is seen that both the experimental and the calculated coordinates of x_{CE} are near x_{GCE} and move slightly forward with increasing AWA. Unfortunately, there is a wide scatter in the experimental values of z_{CE} . This is thought to be because the measured Ks moment contains a large component from the mass of the dynamometer frame and rigging (659kg). This moment was subtracted from the measurement, taking into account the measured heel angle. If there is a slight error in the position of center of gravity of the dynamometer frame, or in the measured heel angle, the error in the calculated moment will be large. However, though there is a scatter in the measured data, it can be seen that z_{CE} is decreasing as AWA increases. The trends in the movement of both x_{CE} and z_{CE} as functions of AWA might be caused by the decrement of force acting on the aft and upper parts of the sails due to the loosening of main and jib sheets with increasing AWA. The calculated results for z_{CE} obtained using the RANS-based CFD show the same trend as the experiments. On the other hand, the calculated results using VLM are considerably higher than the experimental ones. This might be caused by over estimation of the force acting on the upper portion of the mainsail. In this area, since the jib is not overlapping, flow separation may occur easily. However, the VLM does not take flow separation into account.

Figures 14(1) and 14(2) show the calculated results of the sail surface pressure and streamlines using RANS- based CFD. Fig.14(1) indicates the case of experiment ID 96092335 (AWA= 30.7deg.), and 14(2) indicates ID 96080248 (AWA= 37.9deg.). These data correspond to the plotted points on the vertical dotted lines (1) and (2) in Fig.13. In Fig.14, the left and right diagrams correspond to the port and starboard sides, i.e., pressure and suction sides, respectively. In 14(1), although slight flow separation on the suction side of mainsail is seen, the streamlines of both sides run smoothly. On the other hand, in 14(2), considerable flow separation is occurring, in particular, on the suction side of jib. This is the main reason for the reduction of C_L value in the RANS-based CFD calculation at (2) in Fig.13(a). The shapes and three-dimensional coordinates of the sails are given in Table 2. The numbered (1) and (2) tables correspond to the cases of experiment ID 96092335 and ID 96080248, respectively. These also correspond to the calculated results shown in Fig.14. The figures described above the tables show the sail section profiles at 0, 20, 40, 60 and 80% of the sail height. The dimensions of these three-dimensional coordinates are given in the tables including 100% height section data. The positive direction of the *x* coordinate is aft. The four lines at the top of the tables are the measured values for the wind and sail trim conditions, the boat attitude and the sail performance coefficients.



Fig. 14. (1) Surface pressure and streamlines obtained by RANS-based CFD at experimental ID 96092335 (AWA=30.7 deg.) and (2) ID 96080248 (AWA=37.9 deg.).

5.3.2 Variation with mainsail twist angle

Fig.15 shows the performance variation for the mainsail and 130% jib configuration as a function of mainsail twist angle. The mainsail twist was changed by varying the main sheet tension. The boom angle was kept parallel with the boat centerline by moving the main sheet traveler. The experiment was performed for an average value of AWA of 30 ± 2 degrees and mean draft at around 10%. The jib shape was fixed. The twist angle is defined as the angle between the boom line and section chord line at 80% height. In the figure, the

twist angle ranges from 4.5 degrees to 24.9 degrees for the port tack. Varying the twist angle by 20.4 degrees, results in the value of C_X in Fig. 15(b) changing from 0.33 to 0.39 (18%), and the value of C_Y changing from 1.16 to 1.39 (20%). It can be seen that the maximum C_X occurs at a twist angle of around 15 degrees. The considerable decrease in C_Y with increasing twist angle is also worth noticing. In this case, the calculated results of both VLM and RANS-based CFD for C_X and C_Y , and C_L and C_D correspond to the measured values very well.



Fig. 15. Performance variation as a function of mainsail twist angle for mainsail and 130% jib.

Figures 16(1) and 16(2) show the calculated results using RANS-based CFD. Fig.16(1) corresponds to ID 97072213 (twist angle = 8.2 deg.), and 16(2) corresponds to ID 97072218 (twist angle = 24.1 deg.). It can be seen in Fig.16(1) that the streamlines on the upper part of the suction side of the mainsail for the smaller twist angle, show considerable flow

separation. This is caused by the large angle of attack at the upper part of the sail due to the small twist angle. On the other hand, for the higher twist angle shown in Fig.16(2), there is a low negative pressure area at the luff on the suction side of mainsail due to the small angle of attack. This is what causes the considerable reduction in the calculated value for C_X at (2) in Fig.15(b). Table 3 shows the shapes and three-dimensional coordinates of the sails for cases (1) and (2), which correspond to the calculated results shown in Fig.16. Further measured data and comparison with the numerical calculations are described by Masuyama et al. (2007 and 2009)

	(1)	970722	13	AWS m/r	HEEI [dag]	VB	1.5	(2)	970722	18	AWSIm	HEEI Idaal	VB
ť	30.7	8.2	10.5	7.3	16.8	5.1		31.1	24.1	10.6	7.2	12.3	5.1
1	C	Co	Ċ,	Cy	XCE [m]	$z_{GE}[m]$		C,	Co	Cx	C,	X _{CE} [m]	ZOF [m]
T	1.36	0.38	0.37	1.37	0.79	5.96	1	1.22	0.36	0.33	1.23	0.78	5.47
% of		130%Jib			Mainsail	1	% of		130%Jib		1.2.2.2.2.2	Mainsail	
heit	x	y	2	×	Y	2	heit	×	y	2	x	Y	2
	-3.780	0.000	0.000	0.046	0.000	1,320		-3.780	0.000	0.000	0.046	0.000	1.320
- H	-2.817	0.170	0.000	0.934	0.000	1.320		-2.817	0.170	0.000	0.934	0.000	1.320
0	-1.854	0.340	0.000	1.822	0.000	1.320	0	-1.854	0.340	0.000	1.822	0.000	1.320
5	-0.891	0.509	0.000	2.710	0.000	1.320	4	-0.891	0.509	0.000	2.710	0.000	1.320
	0.073	0.679	0.000	3.598	0.000	1.320		0.073	0.679	0.000	3.598	0.000	1.320
1.5	1.036	0.849	0.000	4.486	0.000	1.320		1.036	0.849	0.000	4.486	0.000	1.320
	-2.998	0.000	2.140	0.133	0.000	3.820		-2.998	0.000	2.140	0.133	0.000	3.820
	-2.320	0.440	2.140	0.884	0.214	3.820		-2.321	0.439	2.140	0.852	0.328	3.820
20	-1.595	0.727	2.140	1.638	0.345	3.820	20	-1.598	0.728	2.140	1.584	0.539	3.820
5	-0.839	0.917	2.140	2.395	0.362	3.820		-0.844	0.914	2.140	2.334	0.601	3.820
	-0.067	1.055	2.140	3.156	0.283	3 820		-0.073	1 048	2.140	3.097	0.548	3,820
-	0.714	1,165	2,140	3.919	0.125	3.820		0.708	1.151	2.140	3.866	0.437	3.820
	-2.215	0,000	4.280	0.221	0.000	6.320	121	-2.215	0.000	4.280	0.221	0.000	6.320
	-1.774	0.427	4.280	0.832	0.263	6.320		-1.760	0.403	4.280	0.777	0.378	6.320
40	-1.289	0,746	4,280	1.449	0.42/	6.320	40	-1.266	0.707	4.280	1.358	0.638	6.320
2	-0.750	0,933	4.280	2.076	0.452	6.320	*	-0.729	0.898	4.280	1.977	0.730	6.320
	-0.1/4	1.031	4.280	2/10	0.358	6.320		-0.151	0.985	4.280	2.617	0.721	0.320
-	0.421	1.082	4.280	3.348	0.199	0.320		0.444	1.025	4.280	3.268	0.000	0.020
1.1	-1.433	0.000	6.420	0.306	0.000	8.620	12	-1.433	0.000	6.420	0.308	0.000	0.020
20	-0.011	0.330	6,420	1 222	0.231	0.020	80	-0.002	0.313	6.420	0.701	0.320	0.020
	-0.572	0.390	6 420	1,202	0.372	9.920		-0.544	0.303	6.420	1.590	0.302	9.920
21	-0 100	0.816	6 420	2 108	0.355	8.820	2	-0 168	0.724	E 420	2 068	0.710	9,920
	0 196	0.855	6 420	2687	0 235	8 820		0 229	0.858	6 420	2 568	0.794	8 820
-	-0.650	0.000	8 560	0.396	0.000	11 320	1.00	-0.650	0.000	8 560	0.396	0.000	11.320
	-0.543	0176	8 560	0.662	0 130	11 320	80	-0.533	0 166	8 560	0.627	0 179	11.320
80	-0.416	0.322	8.560	0.932	0.224	11.320		-0.401	0.310	8,560	0.868	0.335	11.320
3	-0.255	0.414	8.560	1,212	0.259	11.320	3	-0.243	0.411	8.560	1,127	0.453	11.320
	-0.070	0.472	8.560	1.499	0.241	11.320		-0.056	0.467	8.560	1.406	0.525	11.320
	0.128	0.510	8.560	1.789	0.197	11.320		0.146	0.500	8.560	1.694	0.576	11.320
	0.132	0.000	10.700	0.483	0.000	13,820		0.132	0.000	10.700	0.483	0.000	13.820
	0.145	0.016	10,700	0.512	0.007	13.820		0.145	0.016	10.700	0.509	0.015	13.820
100	0.161	0.028	10,700	0.541	0.014	13.820	100	0.161	0.028	10.700	0.535	0.031	13.820
5	0.177	0.041	10.700	0.571	0.019	13.820	4	0.177	0.040	10.700	0.561	0.045	13.820
	0.193	0.052	10.700	0.601	0.023	13.820	100	0.193	0.052	10.700	0.588	0.057	13.820
1.00	0.212	0.060	10.700	0.630	0.027	13.820	1.1.1	0.212	0.060	10,700	0.615	0.070	13.820

Table 3. Sail shapes, measured experimental data and three-dimensional coordinates of the sails for the cases of (1) 97072213 and (2) 97072218.





Fig. 16. (1) Surface pressure and streamlines obtained by RANS-based CFD at experimental ID 97072213 (twist angle= 8.2 deg.) and (2) ID 97072218 (twist angle= 24.1 deg.).

6. Discussion of numerical calculation methods

The flow is dominated by multiple-lifting-surface aerodynamic interactions. For larger AWA values, in particular, a large-scale flow separation exists on the leeward side of the sails. In general, there is complex vortex generation in the wake, especially near the top and bottom of the sails, i.e., tip vortices are generated and are influenced by the boundary layer flows on the sails. The resultant aerodynamic forces are mostly dominated by the pressure component, whereas the contribution of the frictional component is generally small. The accurate prediction of the boundary layer flows on the sails and the three-dimensional flow separation, associated with the abovementioned vortex generation, are big challenges for

RANS-based CFD. The geometrical complexity is also another significant challenge to RANS-based CFD. The accuracy in the prediction of the CE is of great interest, in association with the correct prediction of the above-mentioned three-dimensional flow separation.

Through the analyses of the multiblock RANS-based CFD, it appears that the overall trends of the flow and the aerodynamic forces measured in the experiments are fairly well predicted by the present computations. It is also seen that the multiblock domain decomposition considered here is very effective for the present mainsail and jib configurations. The automatic gridding scheme used successfully generates high-quality structured grids for the various sail geometries, AWA, and heel angles considered in the present study. Although there are advantages to a structured grid system for highresolution in the boundary layer flow, building a grid in this fashion is difficult to apply to complex geometries. This problem appears to be resolved by the present scheme.

The Vortex Lattice method is, on the other hand, a convenient tool to predict the lift and induced drag acting on the sail accurately at the apparent wind angles less than about 35 degrees. The computational time of the method is about a few minutes for one calculation condition. The longitudinal coordinates, or x coordinate, of the center of effort of the sail can also be calculated with accuracy by the Vortex Lattice method, however, the estimated vertical coordinate, or z coordinate, of the center of effort by the Vortex Lattice method is considerably higher than the experimental ones. This may be caused by the fact that the flow at the upper portion of mailsail is easily separated because of the absence of jib overlapping, while the flow separation cannot be taken into account in the Vortex Lattice method

7. Conclusion

The sail performance analysis of sailing yacht was carried out by using numerical calculations and experiments. Focus in the present manuscript is especially on the upwind sailing condition. The sails considered here are IMS type, and the shapes and performance were measured by using the sail dynamometer boat *Fujin*. The measured sail flying shapes were used by numerical analysis, where two CFD methods developed by the authors were used, i.e. a multiblock RANS-based CFD method by Tahara and a VLM-based CFD method by Fukasawa. It appears that the overall trends of the flow and the aerodynamic forces measured in the experiments are fairly well predicted by the present computations; and at the same time, the present sail performance database based on the full scale onboard measurements are very useful for validation study of numerical methods. As compared to maturity of VLM, that for RANS-based CFD is still in underway but the future prospect is shown promising, especially for capability in predicting separation flow filed where viscous effects of fluid are significant. The authors believe that our sail performance database associated with accurate sail flying shape measurements will be able to contribute to the further development of more advanced CFD methods.

Although details are not described in the present manuscript, our current effort is directed toward the more challenging problem, i.e., extension of the present work for the downwind sailing condition. Since the onboard sail shape measurement system of *Fujin* is incompetent for the spinnaker measurement due to its balloon shape, the sail shapes and performance are measured using wind tunnel equipment, and such activities are already in progress. The sail shapes are recorded using digital cameras and processed to obtain 3D coordinates using solid shape analyzer software, which provides 3D coordinates from digital photographs taken from several different directions. Importantly, the sail shapes and the sail forces and

moment acting on the model are measured simultaneously. The numerical simulation by using a RANS-based CFD method is also in progress. Along with integration with the aforementioned sail design and performance prediction software *AAF*, more details of our work will be reported in our future publications.

8. Acknowledgments

The sail dynamometer boat *Fujin* was built for sail tests for the Japanese America's Cup entry by a Grant-in-Aid from the Nippon Foundation and the authors would like to thank the Nippon Foundation for providing them this invaluable tool. The authors wish to express their thanks to Yamaha Motor Co. Ltd. for building *Fujin* and to North Sails Japan Co. for making the sails. We would like to thank Dr. Martin Renilson for his valuable discussions and comments on this article. We would also like to thank Mr. H. Mitsui, the harbormaster of the Anamizu Bay Seminar House of the Kanazawa Institute of Technology, for his assistance with the sea trials. Help with the sea trials given by graduate and undergraduate students of the Kanazawa Institute of Technology is also gratefully acknowledged. The graduate students were Masaya Miyagawa, Takashi Hasegawa, and Munehiko Ogihara. Finally, we would like to acknowledge the effort by Mr. Naotoshi Maeda who carried out most of the RANS-CFD simulations as a part of his work on M.S. thesis at Osaka Prefecture University.

9. References

- Baldwin, B.S. & Lomax, H. (1978). Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper, 78-257, AIAA 16th Aerospace Science Meeting, Reno, NV., USA, 1978
- Falkner, V.M. (1943). The Calculation of Aerodynamic Loading on Surface of Any Shape, *Aeronautical Research Committee Reports and Memoranda No.1910, Ministry of Aircraft* Production, London, UK
- Falkner, V.M. (1946). The Accuracy of Calculations Based on Vortex Lattice Theory, British Aeronautical Research Council, No.9621
- Fukasawa, T. & Katori, M. (1993). Numerical Approach to Aeroelastic Responses of Three-Dimensional Flexible Sails, *Proceedings of 11th Chesapeake Sailing Yacht Symposium*, pp.87-105, Annapolis, USA, January 29-30, 1993
- Fukasawa, T. (1993). Aeroelastic Transient Response of 3-Dimensional Flexible Sail, Proceedings of International Conference on Aero-Hydroelasticity, pp.57-62, Beijing, China, October 18-21, 1993,
- Hansen, H., Jackson, P. & Hochkirch, K. (2003). Comparison of Wind Tunnel and Full-scale Aerodynamic Sail Force, *International Journal of Small Craft Technology* (IJSCT), Vol. 145 Part B1, (2003), pp. 23-31
- Hochkirch, K. & Brandt, H. (1999). Fullscale Hydrodynamic Force Measurement on the Berlin Sailing Dynamometer, *Proceedings of SNAME 14th Chesapeake Sailing Yacht Symposium*, pp.33-44, Annapolis, Maryland, USA, January, 1999
- Katori, M. (2009). Advanced Aero Flow Software Manual, North Sails Japan, December 2009.
- Krebber, B. & Hochkirch, K. (2006). Numerical Investigation on the Effects of Trim for a Yacht Rig, *Proceedings of 2nd High Performance Yacht Design Conference*, pp.13-21, Auckland, New Zealand, February, 2006

- Masuyama, Y. & Fukasawa T. (1997a). Full Scale Measurement of Sail Force and the Validation of Numerical Calculation Method, *Proceedings of SNAME 13th Chesapeake Sailing Yacht Symposium*, pp. 23-36, Annapolis, Maryland, USA, January, 1997
- Masuyama, Y., Fukasawa T. & Kitasaki T. (1997b). Investigations on Sail Forces by Full Scale Measurement and Numerical Calculation (Part 1: Steady Sailing Performance), *Journal of Society of Naval Architects of Japan*, Vol. 181, (June 1997), pp.1-13, ISSN 0514-8499, (in Japanese)
- Masuyama, Y., Tahara, Y., Fukasawa, T. & Maeda, N. (2007). Database of Sail Shapes vs. Sail Performance and Validation of Numerical Calculation for Upwind Condition, *Proceedings of SNAME 18th Chesapeake Sailing Yacht Symposium*, pp.11-31, Annapolis, Maryland, USA, March, 2007
- Masuyama, Y., Tahara, Y., Fukasawa, T. & Maeda, N. (2009). Database of Sail Shapes versus Sail Performance and Validation of Numerical Calculation for the Upwind Condition, *Journal of Marine Science and Technology*, JASNAOE, Vol. 14, No. 2, (2009), pp.137-160, ISSN 0948-4280
- Milgram, J. H., Peters, D. B. & Eckhouse, D.N., N. (1993). Modeling IACC Sail Forces by Combining Measurements with CFD, *Proceedings of SNAME 11th Chesapeake Sailing Yacht Sympposium*, pp.65-73, Annapolis, Maryland, USA, January, 1993
- Pistolesi, E. (1937). Betrachtungen uber die gegenseitige Beeinflussung von Tragflügelsystemen. Gesammelte Vorträge der Hauptversammlung 1937 der LilienthalGesellschaft, pp.214-219
- Tahara, Y. & Stern, F. (1996). A Large-Domain Approach for Calculating Ship Boundary Layers and Wakes for Nonzero Froude Number, *Journal of Computational Physics*, Vol. 127, (1996), pp. 398-411
- Tahara, Y. (1995). An Application of Two-Layer k-ε Model to Ship Flow Computation, Journal of Society of Naval Architects of Japan, Vol. 177, (1995), pp. 161-176
- Tahara, Y. (2008). A Reynolds-Averaged Navier-Stokes Equation Solver for Prediction of Ship Viscous Flow with Free Surface Effects, *Proceedings of NAPA User Meeting 2008*, Helsinki, Finland, May, 2008.
- Tahara, Y., Tohyama, S. & Katsui, T. (2006a). CFD-Based Multi-Objective Optimization Method for Ship Design, International Journal of Numerical Methods in Fluids, Vol. 52, (2006), pp. 449-527
- Tahara, Y., Wilson, R., Carrica, P. & Stern, F. (2006b). RANS Simulation of a Container Ship Using a Single-Phase Level Set Method with Overset Grids and Prognosis for Extension to Self-Propulsion Simulator, *Journal of Marine Science and Technology*, Vol. 11, No. 4, (2006), pp. 209-228
- Thompson, J.F., Warsi, Z.U.A., & Mastin, C.W. (1985). Numerical Grid Generation, *Elsevier Science Publishing Co., Inc.,* NY.

ORIGINAL ARTICLE



Propeller rate of revolution (rps)

(N m)

Propeller open water efficiency

Variable decomposition approach applied to multi-objective optimization for minimum powering of commercial ships

Yusuke Tahara¹ · Yasuo Ichinose¹ · Azumi Kaneko¹ · Yoshikazu Kasahara¹

Received: 15 November 2017 / Accepted: 4 April 2018 / Published online: 17 April 2018 © JASNAOE 2018

Abstract

As computational fluid dynamics has matured to the point where it is widely accepted as a key tool for ship hull form design, development of simulation-based design (SBD) has been strongly motivated in the past decades. Although many successful demonstrations of SBD were presented, most cases just deal with minimization of total resistance with a formulation of single-objective optimization problem. Once the interest is in minimization of ship-scale delivered power or effective power, issue related to accuracy of the simulation appears critical in many cases, which yield unconvincing results to hull form designers. The method we propose in this paper aims at overcoming the issues. Instead of just counting on predicted power from the simulation and solve a single-objective optimization problem, we first introduce variable decomposition approach to decompose a target ship performance function into terms including embedded parameters, then formulate and solve a multi-objective optimization problem (MOOP). Any scheme to solve MOOP can be applied. In the following, an overview of the present approach is given and results are presented and discussed through comparison with available experimental fluid dynamics data and detailed analysis of flow and integral parameters. The effectiveness of the present approach is also discussed.

n

Keywords Variable decomposition approach · Multiobjective optimization · RANS-CFD · Energy saving device · Minimum powering · Commercial ship

List of symbols

•			1 1
$L_{\rm PP}$	Length between perpendiculars (m)	V _a	Advance speed of propeller (m/s)
$L_{ m WL}$	Length at the waterline (m)	$J = \frac{V_a}{nD}$	Advance ratio
В	Breadth moulded (m)	T	Propeller thrust (N)
d	Draft moulded (m)	$K_t = \frac{T}{T}$	Thrust coefficient
	Displacement (m ³) Block coefficient	$Q \\ K_{0} = \frac{Q}{m^{2}D^{5}}$	Propeller torque (N m) Torque coefficient
Vs	Ship speed (m/s) Density of water (leg/m^3)	$e^{-\rho n^2 D^3}$	Skin friction correction
ρ g	Gravitational acceleration (m/s ²)	t	Thrust deduction factor, e.g., $t = \frac{T - (R - SFC)}{T - SFC}$
$V Fr = rac{V_{ m S}}{\sqrt{gL_{ m PP}}}$	Kinematic viscosity Froude number	w _n	Nominal wake, $w_{n} = \frac{\int_{0}^{2\pi} \int_{\frac{d_{h}}{2}}^{\frac{D}{2}} urdrd\theta}{\frac{\pi}{2}}$
$Re = \frac{V_{\rm S}L_{\rm PP}}{v}$ R	Reynolds number Total resistance (N)		where the origin is the center of propeller
l+k D	Form factor Diameter of propeller (m)	$w_{\rm e} \\ w_T = rac{V_{\rm S} - V_{\rm a}}{V}$	Effective wake fraction Taylor wake fraction in model scale
Vuentra Taka		w _s	Estimated wake fraction of ship
rusuke ranara		$\mathcal{Q}(\mathcal{O})$	Propener torque in open water

Yusuke Tahara tahara@nmri.go.jp

1 NMRI, National Maritime Research Institute, Mitaka, Japan

🙆 Springer

 $\eta_0 = \frac{TV_{\rm a}}{2\pi n Q(O)}$

1 Introduction

As computational fluid dynamics (CFD) has matured to the point where it is widely accepted as a key tool for ship hull form design, development of simulation-based design (SBD) has been strongly motivated in the past decades. The most complete survey of such work is found in ITTC reports presented in every 3 years, including CFD-based hull form optimization in seakeeping/manoeuvring as well as resistance/propulsion research fields [1-3] including related work of interest [4–11]. Additionally, some recent examples found in the present journal are Kim et al. [12] and others [13–16]. The SBD was proposed in many forms, where high performance computing CFD solvers, optimization algorithms, geometry and grid manipulation automatic methods are effectively integrated. Focus of the present work is also there. SBD developed in the previous work was successfully used in the optimization of high speed monohull, multihull, and displacement ships (Tahara et al. [17-19], Campana et al. [20], Kandasamy et al. [21], Diez et al. [22]). Our new challenge is to achieve the next goal of SBD in commercial ship design, i.e., advanced-level SBD is coupled with traditional design arts to fully utilize the expertise and at the same time to consider newly introduced ship design index in International Maritime Organization (IMO), i.e., Energy Efficiency Design Index (EEDI) or Energy Efficiency Operational Indicator (EEOI); which background also yields a modern design trend to install optimal energy saving devices (ESD, see Fig. 1 for example).

In such applications of SBD to ship design, accurate prediction of powering parameters, e.g., self-propulsion factors (i.e., t, w_t , and η_R), is especially important, while issues related to scaling of ESD effects must be carefully treated [23]. In fact, the traditional design looks into details of embedded parameters related to powering, and in practise, solves multidisciplinary or multi-objective problem, where the parameters are sometimes conflicting, i.e., the improvement of a specific parameter causes the worsening for some others. Nevertheless, most of simulation-based optimizations reported in the past decades are to minimize the total resistance, and in some cases, in associated with other ship performance parameters, e.g., seakeeping merit function. We recently investigated the CFD capability to predict the parameters, observing the related workshop data (CFD workshops: Gothenburg 2010



Fig. 1 Overview of stern geometry for 82BC test case. Both rudder and ESD are attached to the bare hull

[24], Tokyo 2015 [25]) and data from EEDI test cases by using NMRI CFD codes, and have concluded that, in general, powering parameters are fairly well-predicted with accuracy in trends, but the quantitative accuracy needs more improvement especially for w_t (Kasahara [26], Ichinose et al. [27]). Consequently, accuracy in the predicted power, i.e., delivered power (DP) or effective power (EP), totally depends on the quantitative accuracy of the selfpropulsion factors, and in many cases, this fact yields unconvincing results to hull form designers. As far as the trends are trustworthy, an appropriate approach must be introduced to make fully use of current CFD capability for hull form design.

The method we propose in this paper aims at overcoming the above-mentioned issues. Instead of just counting on predicted power from the simulation and solve a singleobjective optimization problem (SOOP), we first introduce separation of variable decomposition approach (VDA) to decompose a target ship performance function into terms including embedded parameters, then formulate and solve a multi-objective optimization problem (MOOP). Any scheme to solve MOOP can be applied. Indeed, derived equation in VDA form is shown very useful to evaluate impact of each factor on a target function. And as far as this is solved in a form of MOOP by using the present optimization scheme, quantitative accuracy to evaluate each objective function may differ; where in this regard, the approach for MOOP to deal with different accuracy (or fidelity) CFD was successfully demonstrated and examined in the previous work [18]. The optimization will yield Pareto optimal set when the objective functions are conflicting relation, and impact of each function on target function can be separately corrected by using design experience. Or, the estimation given in the present VDA form may appear to be reasonably accurate. As it is shown later, letting the delivered power be a target

261

function, we have VDA form involving terms related to selfpropulsion factors and that is quite suitable to account for the traditional design intuition.

Our approach is demonstrated by using NMRI-SBD, an extended version of the earlier work for more capability and practical applications. For instance, that takes over the threecomponent-SBD model, which is composed of optimizer, geometry modeler, and asynchronous CFD evaluator modules. Each of those controls functionally independent and replaceable submodules. Methods to perform CFD along with preprocessing including meshing and post-processing are NMRI in-house codes, i.e., HULLDES, AUTODES, GTOOL, NEPTUNE, SURF and NAGISA (NMRI CFD 2015 [28]), which are capable to use high-efficiency single-block and more flexible multi-block computational grids, where the latter is essential feature to perform ESD optimization. The CFD methods are implemented into a self-propulsion simulator, where the Reynold-averaged Navier-Stokes (RaNS) solver is coupled with a propellerperformance programme in an interactive and iterative manner. The yielded powering parameters are used in the present optimization in a form of multi-objective functions derived by the aforementioned VDA. The test ship models considered here are NMRI original design and contractor design bulk carriers and tankers, for all of which experimental fluid dynamics (EFD) data are available for validation. In the following, an overview of the present approach is given and results are presented and discussed including comparison with available EFD data and detailed analysis of flow and integral parameters. The effectiveness of the present approach is also discussed. Finally, conclusion is given.

2 Overview of VDA: analysis and definition of objective function

In this section, an overview is given of method we propose, i.e., VDA to analyze the target function and redefine the multi-objective functions to be minimized. A background is our recent investigation on the CFD capability to predict powering parameters. The related workshop data (Gothenburg 2010 [24], Tokyo 2015 [25]) and data from EEDI test cases using NMRI CFD codes are carefully examined. For example, Fig. 2 shows the comparison of ship resistance between CFD and EFD for a NMRI chemical tanker test case, for which accuracy in prediction of wave-making resistance is relatively important at the design speed (i.e., Froude number ~ 0.18). Figure 3 shows the comparison of nominal wake between CFD and EFD for a NMRI bulk-carrier test case. It is seen that ship resistance along with wake fields in towing condition is generally well predicted by most of CFD methods; although higher order turbulence model indicates better agreement as demonstrated for (b). However, as shown in Fig. 4 for a NMRI



Fig. 2 Comparison of total resistance between CFD and EFD. 33CT test case for model scale condition. NAGISA is used with 2.8 Mil. cells

bulk-carrier test case as an example, we still observe difficulties in self-propulsion condition, i.e., powering parameters are fairly well-predicted with accuracy in trends, but the quantitative accuracy needs more improvement. Since accuracy in the predicted power totally depends on those, this fact must be considered in development of more reliable next-generation SBD.

Our proposed approach is as follows: instead of just counting on predicted power from the simulation and solve a SOOP, we first introduce VDA to decompose a target ship performance function into terms including embedded parameters, then formulate and solve MOOP. In the following, details are given of our VDA, where, the target function is delivered power (DP), which will be the most straightforward example. Two cases, i.e., a simplified form and a more general form, are presented.

2.1 Case 1: simplified form

First, simplified form of DP is derived to demonstrated and show overview of the present VDA. By definition, DP is given as,

$$DP \propto \frac{RV_{\rm S}}{\eta_{\rm P}}.$$
(1)

Here, *R* is ship resistance, *Vs* is ship speed, and η_P is propeller efficiency. During the optimization, assuming that *Vs*, thrust deduction coefficient 1 - t, propeller efficiency ratio η_R are constant, effective wake $1 - w_e$ is proportional to nominal wake $1 - w_n$, i.e., $1 - w_e \propto 1 - w_n$ (= W_n), advance ratio $J \propto W_n/nD$ (*n*: number of propeller rotation, *D*: propeller diameter), and engine control is for constant torque. Besides, local linearization of variables enables to write Eq. 1 as,

$$\frac{\delta \mathrm{DP}}{\mathrm{DP}} = \frac{\delta R V}{R} - \frac{\delta \eta_{\mathrm{P}}}{\eta_{\mathrm{P}}}.$$
(2)

Fig. 3 Nominal wake distribution for 82BC test case for 7 m model \blacktriangleright scale condition. **a** Neptune HO Grid 1.46 Mil. Cells (both sides) with MSA turbulence model, **b** Nagisa OO Grid 2.46 Mil. Cells (both sides) with $k - \omega$ EASM turbulence model, and **c** measurements. See Ref. [30] for more detailed conditions

Also from definition,

$$\eta_{\rm P} = \frac{K_t}{K_{\rm q}} \frac{J}{2\pi} \frac{1-t}{1-w_{\rm e}} \eta_R,\tag{3}$$

$$T = \rho n^2 D^4 K_t = \frac{R}{1-t},\tag{4}$$

where K_t and K_q are trust and torque coefficients, ρ is fluid density. In similar manner to Eq. 2, the following are given by definition of η_P and R;

$$\frac{\delta\eta_{\rm P}}{\eta_{\rm P}} = \frac{\delta K_t}{K_t} + \frac{\delta J}{J} - \frac{\delta K_{\rm q}}{K_{\rm q}} - \frac{\delta W_{\rm n}}{W_{\rm n}},\tag{5}$$

$$\frac{\delta R}{R} = \frac{\delta K_t}{K_t} + \frac{2\delta n}{n} + \frac{4\delta D}{D}.$$
(6)

Additionally, definition of J and Eq. 6 give:

$$\frac{\delta J}{J} = \frac{\delta W_{\rm n}}{W_{\rm n}} - \frac{\delta n}{n} - \frac{\delta D}{D}$$

$$= \frac{\delta W_{\rm n}}{W_{\rm n}} - \frac{1}{4} \left(\frac{\delta R}{R} - \frac{\delta K_t}{K_t} + \frac{2\delta n}{n} \right)$$

$$= \frac{1}{1 - \alpha_t / 4} \left(\frac{\delta W_{\rm n}}{W_{\rm n}} - \frac{\delta R}{4R} - \frac{\delta n}{2n} \right).$$
(7)

Assuming that the following parameters are given by design experience,

$$\alpha_t = \frac{\delta K_t}{\delta J} \frac{J}{K_t}, \ \alpha_q = \frac{\delta K_q}{\delta J} \frac{J}{K_q}.$$
(8)

Hence, we have an expression of DP as follows:

$$\frac{\delta \text{DP}}{\text{DP}} = \frac{\delta R}{R} - \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4} \left(\frac{\delta W_n}{W_n} - \frac{\delta R}{4R} - \frac{\delta n}{2n} \right) + \frac{\delta W_n}{W_n}.$$
(9)

Furthermore, torque constant control is considered to rewrite the term related to n, which yields

$$\frac{\delta \text{DP}}{\text{DP}} = \left(1 + \frac{1}{4} \left(\frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right)\right) \frac{\delta R}{R} + \left(1 - \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right) \frac{\delta W_n}{W_n} + \frac{1}{2} \left(\frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}\right) \frac{\delta \text{DP}}{\text{DP}}.$$
(10)







Deringer



Fig. 4 Comparison of self-propulsion factors for 82BC test case for fully appended condition. Unstructured grid CFD code SURF is used with meshing code HEXPRESS [29]. The total number of cells varies in 5.76–7.84 Mil. depending on duct geometry. WAD0 through WAD4 correspond to NMRI WAD test models with large variety of geometry. See [31] for more detailed information

Here, coefficients are redefined and a simpler form can be given by,

$$\frac{\delta \text{DP}}{\text{DP}} = \frac{\alpha_1}{\alpha_0} \frac{\delta R}{R} + \frac{\alpha_2}{\alpha_0} \frac{\delta W_{\text{n}}}{W_{\text{n}}},\tag{11}$$

where

$$\begin{cases} \alpha_0 = 1 - \frac{\alpha_3}{2}, \ \alpha_1 = 1 + \frac{\alpha_3}{4}, \ \alpha_2 = 1 - \alpha_3 \\ \alpha_3 = \frac{1 + \alpha_t - \alpha_q}{1 - \alpha_t/4}. \end{cases}$$
(12)

Finally, an approximated form of DP using Eq. 12 is

$$DP \propto R^{\alpha_1/\alpha_0} W_n^{\alpha_2/\alpha_0}$$
(13)

or

$$DP^{\alpha_0/\alpha_1} \propto RW_n^{\alpha_2/\alpha_1} = RW_n^{\alpha}.$$
 (14)

For simplicity, Eq. 14 can be used for definition of objective function to be minimized, i.e.,

$$\operatorname{Min} F = RW_{\mathrm{n}}^{\ \alpha},\tag{15}$$

which was investigated by the authors for a single-objective optimization test case in earlier work [32]. This problem can be solved as multi-objective optimization problem, defined as

$$\operatorname{Min.} \begin{cases} F_1 = R\\ F_2 = W_n \end{cases}$$
(16)

where coefficients in Eq. 11 are positive.

2.2 Case 2: more general form

Next, a more general form of DP is derived. Again, based on definition, DP is given by

$$DP \propto \frac{RV_{\rm S}}{\eta_{\rm P}} = RV_{\rm S} \frac{1 - w_{\rm S}}{1 - t} \frac{1}{\eta_0 \eta_R} = \frac{TV_{\rm S}(1 - w_{\rm S})}{\eta_0 \eta_R}.$$
 (17)

 η_0 is given by using ideal efficiency, i.e.,

$$\eta_0 = \alpha_{\rm P} \frac{2}{1 + \sqrt{1 + C_{\rm th}}},\tag{18}$$

where constant $\alpha_{\rm P}$ (≈ 0.65 for MAU and 0.75–0.85 for conventional propellers) is related to momentum and viscous losses. Introducing propeller advance speed $V_{\rm a}$, thrust coefficient $C_{\rm th}$ is defined as

$$C_{\rm th} = T \left/ \left(\frac{\rho V_{\rm a}^2}{2} \frac{\pi D^2}{4} \right)$$
(19)

all of which yields alternate expression of DP as follows:

DP
$$\propto \frac{TV_{\rm S}(1-w_{\rm S})}{V_{\rm a}\eta_{\rm P}} \frac{1+\sqrt{1+C_{\rm th}}}{2\alpha_{\rm P}} \frac{1}{\eta_{\rm R}}$$
 (20)

or

$$DP \propto \frac{\rho \pi D^2}{16 \alpha_P \eta_R} V_a^{\ 3} C_{th} (1 + \sqrt{1 + C_{th}}). \tag{21}$$

Introducing approximation form for the term related to $C_{\rm th}$ in the following manner,

$$C_{\rm th}(1+\sqrt{1+C_{\rm th}}) \approx \beta C_{\rm th}^{\ r}.$$
(22)

🙆 Springer

-529-

Then, DP can be expressed in more practical form as below:

$$DP \propto \frac{\rho \pi D^2}{16 \alpha_{\rm P} \eta_R} V_{\rm a}^{\ 3} \beta C_{\rm th}^{\ r}$$

$$= \frac{8^r (\rho \pi D^2)^{1-r} \beta V_{\rm S}^{\ 3-2r}}{16 \alpha_{\rm P} \eta_R} (1 - w_{\rm S})^{3-2r} R^r (1 - t)^r$$
(23)

or

DP
$$\propto \frac{1}{\eta_R} (1 - w_S)^{3-2r} R^r (1 - t)^r.$$
 (24)

Note that a chart shown in Fig. 5 or any design chart/ empirical formula can be used to derive Eq. 19. In our experience using Fig. 5 for the full ships with $C_{\rm th}$ =0.75–0.85, we have β =2.23 and r=1.23 indicating within 0.5% approximation error in Ee.22 (i.e., differences between right and left side terms), which is considered to be sufficiently accurate. Finally, the following expression of DP is derived as demonstrated earlier,

$$\frac{\delta DP}{DP} = r\frac{\delta R}{R} - r\frac{\delta(1-t)}{(1-t)} + (3-2r)\frac{\delta(1-w_{\rm S})}{(1-w_{\rm S})} - \frac{\delta\eta_R}{\eta_R}$$
(25)

or

$$\frac{1}{r}\frac{\delta DP}{DP} = \frac{\delta R}{R} - \frac{\delta(1-t)}{(1-t)} + \frac{3-2r}{r}\frac{\delta(1-w_{\rm S})}{(1-w_{\rm S})} - \frac{1}{r}\frac{\delta\eta_R}{\eta_R}.$$
 (26)

For r = 1.23 it yields,

$$\frac{1}{r}\frac{\delta \mathrm{DP}}{\mathrm{DP}} \approx \frac{\delta R}{R} - \frac{\delta(1-t)}{(1-t)} + 0.433\frac{\delta(1-w_{\mathrm{S}})}{(1-w_{\mathrm{S}})} - 0.811\frac{\delta \eta_{R}}{\eta_{R}},$$
(27)



Fig.5 Efficiency of various propulsion devices and C_{th} ranges for ship classes (Breslin et al. 1994 [33])

which agrees well with our design experience. Using similar representation of the previous section, this problem where DP is minimized can be solved as multi-objective optimization problem, defined as

Min.
$$\begin{cases} F_1 = R \\ F_2 = -(1 - t) \text{ or } F_2 = t \\ F_3 = 1 - w_s \\ F_4 = \eta_R \end{cases}$$
 (28)

Furthermore, our unit analysis in which ESD is considered to be a part of propulsion system considers $\delta R = 0$, hence we have,

$$\frac{1}{r}\frac{\delta DP}{DP} = -\frac{\delta(1-t)}{(1-t)} + \frac{3-2r}{r}\frac{\delta(1-w_{\rm S})}{(1-w_{\rm S})} - \frac{1}{r}\frac{\delta\eta_R}{\eta_R}$$

$$\approx -\frac{\delta(1-t)}{(1-t)} + 0.433\frac{\delta(1-w_{\rm S})}{(1-w_{\rm S})} - 0.811\frac{\delta\eta_R}{\eta_R},$$
(29)

and if
$$\delta(1 - w_{\rm S})/(1 - w_{\rm S}) \approx \delta(1 - w_t)/(1 - w_t)$$
 is valid
 $\frac{1}{r} \frac{\delta \text{DP}}{\text{DP}} \approx -\frac{\delta(1 - t)}{(1 - t)} + 0.433 \frac{\delta(1 - w_t)}{(1 - w_t)} - 0.811 \frac{\delta \eta_R}{\eta_R}.$ (30)

Further assuming that $\delta \eta_R$ is negligible and using similar representation of the previous section, this problem where DP is minimized can be solved as multi-objective optimization problem which defines,

Min.
$$\begin{cases} F_1 = t \\ F_2 = 1 - w_t \end{cases}$$
 (31)

It is noteworthy that the above analysis will provide reasonable estimation of gain of a target function, i.e., in this case DP, together with individual impact of each term on the target function. This feature is evaluated later.

3 Overview of NMRI-SBD method

NMRI SBD consists of the three main components; i.e., optimizer (OPT), geometry modeler (GM), and asynchronous CFD evaluator (AE) modules. Each of those controls functionally independent and replaceable submodules involving deterministic and stochastic/single-objective (SO) and multiobjective (MO) optimization methods, CAD direct/indirect control interface and surface morphing method, and both of the two major CFD approaches, i.e., potential flow based-CFD (PF-CFD) and RaNS based CFD (RANS-CFD). The three main modules can be used in a combined or separated form depending on design needs.
As shown in Fig. 6, the three components are defined in a general manner. The OPT module includes SOOP and MOOP schemes, which covers gradient and non-gradient based schemes. The GM module offers capability in both CAD and morphing based design modification methods. Finally, the AE module is implemented to make flexible interface with both parallel and/or serial mode PF and/or more expensive RANS-CFD. OPT and AE modules are coded based on MPI parallel computing architecture.

3.1 RANS-CFD method

Adopted RANS-CFD methods are all NMRI in-house codes, NEPTUNE, SURF, and NAGISA, which have been well-validated through many NMRI internal and external industrial test cases and international events to competitively evaluate the ship hydrodynamics CFD capabilities (e.g., CFD Workshop 2015 [25]). NEPTUNE and SURF are general-purpose, structured and unstructured grid, respectively, high performance parallel computing RANS codes. The finite volume discretization and pseudo-compressibility velocity–pressure coupling approach are adopted. Several isotropic turbulence models are available, while in the present application, Spalart–Allmaras turbulence model is used [34].

NAGISA is the latest multi-block structured grid code, i.e., extended version of NEPTUNE, including overset grid capability to provide flexibility in grid generation, local grid refinement, and for bodies and/or blocks with relative motions; and $k - \omega$ two-equation-based isotropic and anisotropic turbulence models [35]. For all codes, prediction of sinkage and trim is possible using moving grid approach; and the self-propulsion simulation is also possible using simplified propeller theory along with body-force approach to couple with RANS equations, which effectively enhances computational performance. In the present study, self-propulsion simulation is basically performed for ship-point selfpropulsion condition.



Fig. 6 Basic components and module interfaces of the present NMRI simulation-based design (SBD)

Deringer

Regarding grid study and uncertainty assessments, which relate to the present study, see Kasahara [26], and Ichinose et al. [27], where both in-house and commercial meshing tools described below are used.

3.2 CAD, meshing, and ESD modelling tools

The primary CAD system in the present study is NAPA [36], which is at present one of the most widely used CAD systems in the domestic and foreign commercial ship yards. In addition, RHINOCEROS [37] is used for detailed ESD design, which numerical data are directly converted to 3D printer input, and the new test model is prepared within a day. Along with commercial structured and unstructured meshing tools, i.e., GRIDGEN [38] and HEXPRESS [29], respectively, NMRI in-house codes AUTODES and GTOOL are used. The latter is developed especially for ESD parametric design and meshing, along with overset preprocessing for NAGISA code. The AUTODES is extended version of the HULLDES for parametric hull form modification and GTOOL GUI, and also SQP based automatic optimization by coupling with NMRI CFD codes. For optimization of ESD demonstrated later, parametric ESD modelling feature of GTOOL is used (see [39, 40] for additional information especially related to this application). In addition, blending (morphing) type shape modification and grid re-generation is also used for hull form optimization test case described later in a similar manner demonstrated in the earlier work [19].

3.3 General definition of multi-objective optimization problem

Shape design optimization is typically formulated in the framework of non-linear programming (NLP) problem. For a general expression of *N*-objective function optimization problem in ship hydrodynamics, the mathematical formulation assembles all the design variables $x_1, x_2,..., x_M$ in a vector $\vec{x} = (x_1, x_2,..., x_M)^T$ belonging to a subset χ of the *M*-dimensional real space \Re^M , that is $\vec{x} \in \chi \subseteq \Re^M$ (upper x^{μ}_i and lower x^l_i bounds are typical enforced onto the design variables). The objective of the optimization $\vec{F} = (F_1, F_2,..., F_N)^T$ and the equality and inequality constraints h, g are functions of the design variables \vec{x} and of the state of the system $\vec{u}(\vec{x})$. A general form for constrained NLP problems is then to find the particular vector \vec{x} in the subset χ which solves the following:

$$\operatorname{Min}:\begin{cases} F_1(\vec{x}, \vec{u}(\vec{x}); Re, Fr) \\ F_2(\vec{x}, \vec{u}(\vec{x}); Re, Fr) \\ \cdot \\ \cdot \\ F_N(\vec{x}, \vec{u}(\vec{x}); Re, Fr) \end{cases}$$
(32)

Subject to.
$$\begin{cases} h_{j}(\vec{x}) = 0 \ (j = 1, \dots, p) \\ g_{j}(\vec{x}) \le 0 \ (j = 1, \dots, q) \\ x_{i}^{l} \le x_{i} \le x_{i}^{u} \ (i = 1, \dots, M) \end{cases}$$
(33)

where h_i and g_i are constraint functions.

3.4 Evolutionary algorithm: real-coded multi-objective genetic algorithm

EA adopted in the present study is the real-coded multiobjective genetic algorithm (RC-MOGA). The authors competitively evaluated both RC-MOGA and more classical Binary-coded MOGA (BC-MOGA) in the earlier work. It is found that each approach offers the advantage over the other depending on the problem setup, e.g., if the design variables are given as continuous real number, RC-MOGA is more suitable. The capability RC-GA in ship design optimization is investigated in the authors' earlier work. In more general engineering applications details of both algorithms are discussed by Deb [41]. In addition, it is noteworthy that since the EA uses random coefficients to enhance the dynamic of the individuals exploring the design space, a large campaign of computations should be performed to obtain statistically significant results. Although such an approach is computationally extremely expensive, it provides systematic results and the effectiveness is focused in the present study.

The basic algorithm of RC-MOGA in the present optimization framework is illustrated in Fig. 7. As demonstrated in the previous work, higher fitness f is given to individuals of higher Pareto ranking $R_{\rm P}$, i.e., $f = 1/R_{\rm P}$. As above mentioned, a drawback of evolutionary family

algorithms is increase of the computational load; however, that is overcome in the present framework by introducing parallel computing technique, i.e., message passing interface (MPI) protocol with AE model.

4 Example of application: exploration of hull form design space and investigation of associated flow features

In the following, we discuss detailed evaluation of the present approach associated with practical design projects, involving exploration of its design space and investigation of hull form and associated flow features. Test parent design is two types: 33CT and 82BC hull forms are designed in NMRI EEDI Project Team, while the alphanumeric codes stand for 33,000 DW Chemical Tanker and 82,000 DW Bulk Carrier, respectively. These are designed in NMRI for investigation purpose and the geometry is available for public use. Principal dimensions for the baseline designs are shown in Table 1.

Self-propulsion factors of the baseline designs are evaluated by using the implemented CFD method and the results are compared with EFD data. NEPTUNE is used with zero Fr model scale conditions and ship-point self-propulsion conditions are performed by using separately designed clockwise rotating propellers for 82BC and 33CT test cases.

Table 1Principal dimensionsfor baseline designs for 82BCand 33CT test cases (82BC and33CT correspond to 82K DWTBulk Carrier and 33K DWTChemical Tanker, respectively)

	82BC bulk car- rier	33CT chemical tanker
$L_{\rm wl}$ (m)	225.0	173.0
$L_{\rm pp}$ (m)	222.0	170.5
<i>B</i> (m)	32.26	27.7
<i>d</i> (m)	12.2	10.0
Cb	0.87	0.80
$V_{\rm s}({\rm kn})$	14.2	15.3



Fig. 7 Parallel computing algorithm and asynchronous evaluation for evolutionary optimizer module

Table 2Comparison of CFDand EFD results for 33CT and82BC baselines

	EFD	CFD	Error
33CT			
1+k	1.246	1.208	-3.0%
1-t	0.835	0.831	-0.5%
$1 - w_t$	0.659	0.646	-2.0%
82BC			
1+k	1.310	1.227	-6.3%
1-t	0.842	0.837	-0.6%
$1 - w_t$	0.542	0.582	7.4%

The full-load and even-keel conditions are assumed. Topology and total number of computational grids are HO type and about 1.6 million cells. Table 2 shows the comparison of self-propulsion factors between the CFD and EFD (using 7 m model). It is seen that trend of values in EFD between the two models are accurately predicted by CFD. For more detailed information of grid study, see Ichinose et al. [27]. We prepare the computational grid with relatively small size to avoid significant increase of computational hours in extensive series computations, paying enough care for the accuracy in trends as discussed in the reference. Hence, NMRI SBD is used to explore the design space and observe trend of the target functions derived by VDA. The focus is on the three target functions as follows:

Evaluate
$$\begin{cases} F_1 = R\\ F_2 = t\\ F_3 = 1 - w_t \end{cases}$$
 (34)

where the formulation of the target functions is based on Eq. 28 and $\delta\eta_R$ is assumed negligible. Hence, our approach to explore the design space is: (1) select a type ship from NMRI design database; (2) optimize mainly forebody to minimize wave-making resistance and afterbody to minimize viscous resistance (i.e., F_1), and define an initial baseline design; (3) keeping the same displacement, optimize afterbody to minimize self-propulsion factors, i.e., F_2 and F_3 in Eq. 34, and define sister baseline designs; and (4) create and examine new sibling designs using the hull form blending approach, which operation is written by,

$$P = a_1 P_1 + a_2 P_2 \text{ for } n = 2$$

where
$$\begin{cases} a_1 = \alpha_1 \\ a_2 = (1 - \alpha_1) \end{cases}$$

and
$$\vec{P} = a_1 \vec{P}_1 + a_2 \vec{P}_2 + a_3 \vec{P}_3 \text{ for } n = 3$$

where
$$\begin{cases} a_1 = \alpha_1 \\ a_2 = (1 - \alpha_1)\alpha_2 \\ a_3 = (1 - \alpha_1)(1 - \alpha_2)\alpha_3 \end{cases}$$

So that
$$\sum_{i=1}^{n} a_i = 1,$$

(35)

1

where *n* is number of baseline design to be used in the blending operation, $\vec{P}_1, \vec{P}_2, ..., \vec{P}_n$ are surface points for corresponding n baseline designs; and α_j are design variables. α_j is typically bounded as $0 \le \alpha_j \le 1$ for internal blending and the bound can be extended negative and more-than one for external blending. The computational volume grid is regenerated similarly, by defining $\vec{P}_1, \vec{P}_2, ..., \vec{P}_{n+1}$ to be grid points in solution domain for the baseline designs. For the results discussed below, the variation of design variables are $0 \le \alpha_i \le 1$ and $-0.1 \le \alpha_i \le 1.1$ for internal and external blending, respectively; and uniform random number is used to give α_i .

Optimization here is manually done by designers by moving CAD control points, in aid of design expertise, EFD design database, and CFD analysis. Modification of crosssectional volume distribution is referred to as "UV modification" in our study, which is characterized by shape of framelines and trends in flow; i.e., as compared to V-type framelines, relatively more pronounced U-type framelines





yield larger viscous resistance and stronger bilge vortices. Figures 8 and 9 show the comparison of baseline designs for 82BC and 33CT test cases, respectively; and Figs. 10 and 11 show the comparison of the target functions for 82BC and 33CT test cases, respectively. As shown in the figure, the target functions indicate that F_2 vs. F_3 are apparently conflicting relations for both 82BC and 33CT cases; and F_1 vs. F_3 are also conflicting, but more complexities are shown for 33CT case. In the following, details of our analysis are given.

For BC test case, the initial baseline design is Model A. This model indicates significantly low viscous resistance (i.e., F_1), and resultant stern frameline shape is V-type. On the other hand, $1 - w_1$ (i.e., F_3) of Model A is shown too high, which apparently causes low propulsive performance. Then, Model B is designed by modifying afterbody for Model A, aiming at opposite hydrodynamic trends as much as possible. As shown in Fig. 10, the effort is seen considerably successful, but it also appears that Model B has significantly deteriorated t (i.e., F_2). This suggests redesign of the afterbody to yield Model C which indicates lower t. Both longitudinal and cross-sectional volume distributions are modified so as to lower t. More specifically, the stern end waterline tangential angle near the top of propeller disk is carefully modified, especially observing trends in the surface pressure field predicted by CFD for self-propulsion condition. Finally, aforementioned hull form blending is used to explore the design space, and we found that the present three baseline designs are good parent baseline designs to yield variety of sibling designs.

On the other hand, we perform the similar approach for CT test case. The initial baseline design is New Design 1 shown in the figure. This model is aimed to have low viscous resistance (i.e., F_1) and the resultant stern frameline shape is

also V-type. Then it appears that t (i.e., F_2) of New Design 1 must be lowered, and New Design 2 is developed. In similar manner to that for BC test case, the stern end waterline tangential angle near the top of propeller disk is carefully modified. At this point, we performed blending approach to identify important geometrical and hydrodynamic trends in the design space, while we found that the latter presents quite different trends from those for BC test case. We made further effort to lower F_1 , then developed New Designs 3 and 4, both of which are shown very promising.

The above-mentioned exploration of design space may not be efficiently performed without utilizing the present approach, i.e., SBD with detailed observation of decomposed target parameters. The propulsive factors in this case are theoretically derived by the present VDA. Importantly, the present way to explore design space is very effective to take the traditional design expertise into account, since traditional designers usually looks into details of embedded parameters related to powering, and in practise, solves multidisciplinary or multi-objective problem.

A detailed investigation of trends between geometry and flow is another important objective of the present study. More results for different aspects of flow are also given in Ichinose et al. [27]. In the analysis, the hull form-blending approach presented in Eq. 35 is used between the baseline and final sister designs for both 33CT and 82BC test cases (see Fig. 9 for the geometry). Full scale and model scale are considered. The topology type and size of computational grids for model scale are same as for the previous analyses. Those for full scale are the OO topology NEPTUNE usable and about 2.4 and 2.2 million cells ($321 \times 73 \times 105$ and $321 \times 73 \times 97$, respectively), for 33CT and 82BC test cases, respectively. The minimum spacing normal to wall is set to be $y^+ = 0.75$ for both model- and full scales.



Fig. 10 Exploration of design space (82BC test case)

First, model-scale computational results are discussed. Figure 12 shows comparison of surface pressure and limiting streamlines. On the afterbody, a pressure pocket (low pressure region) is located near the stern bilge, which causes the streamlines to converge towards the stern bilge, and the streamlines meet those from the flat bottom and finally form the stern bilge vortex. It was expected to see the trends in classical discussion, i.e., the U-type frame-line stern form would produce broader and deeper pressure pocket, yielding larger and stronger stern bilge vortex. However, those are slightly different in the present case, i.e., the present U-type stern form produces weaker and backward pressure pockets as compared to the V-type. This is apparently due to the way in the recent design of this type, i.e., a special care is always



Fig. 11 Exploration of design space (33CT test case)

paid for bilge part near the pressure pocket not to be too expand so as to preclude significant increase of the viscous pressure resistance. As modified from V-type to U-type, the slopes of separation lines become steeper according to steepness of contour lines on pressure distributions. These changes of flow field affect to bilge vortices as discussed below.

Figure 13 presents limiting streamlines associated with cross-sectional representation of axial vorticity distributions, where the vorticity contour lines clearly show generation of bilge vortices associated with merging of limiting streamlines. As noted above, depending on steepness of **Fig. 12** Pressure distributions and limiting streamlines. Model-scale condition (left and right for 33CT and 82BC, respectively; top and bottom for V and U-types, respectively)





Fig. 13 Limiting streamlines and axial vorticity distribution. Model-scale condition (left and right for 33CT and 82BC, respectively; top and bottom for V and U types, respectively)

🖄 Springer

-536-

Journal of Marine Science and Technology (2019) 24:260–283





contour lines of pressure distribution and separation lines in the region very close to the stern, axial vorticity for U-type becomes stronger than that of V-type. The results related to viscous pressure resistance are discussed in association with Fig. 14, presenting longitudinal components of normal pressure force acting on the surface, for which integral over the surface yields viscous pressure resistance. The U-type shows expanded negative region that corresponds to the raising of the hollow parts of hull forms. These differences lead to the increase of the viscous pressure resistance; i.e., increases of form factor 1 + k are from V-type to U-type are 2.9 and 6.2% based on U-type for 33CT and 82BC, respectively.

Finally, full scale results are discussed. Figures 15 and 16 show the comparisons of results for 33CT test case for model and full scales. The figure also includes results of potential flow simulation, which are obtained from Hess-Smith type surface panel method approach [42]. There are some expected trends in those flow aspects, e.g., full scale viscous surface pressure distributions show more similarity to those of the potential flow. Additionally, as expected, overall features of surface streamlines are mostly similar between the two, but those near the stern are apparently different due to viscus effects of flow. On the other hand, it may be noteworthy that impacts of the Re difference on the flow differ between the U- and V-types. The difference of form factor 1 + k between U and V types is 2.9% (based on

value for U type) for model scale, and that for full scale is 4.3%. Those implies that three-dimensional flow separation and associated vortex generation and increase in drag occur under more complex influences of *Re* differences between the two scales, and simple scaling methods obviously have limitations to account for those. Of course, the validation of full scale simulation is still in progress and further investigation is necessary to have certain conclusion. At present, results shown here are found very promising, with respect to a fact that CFD will be a key technique to predict full-scale ship performance and this is especially true for future ESD design [23].

In summary, it is shown in the present demonstration that the present SBD approach in association with VDA of target function effectively cooperates with the traditional design. If this is properly used, the knowledge and capability of the traditional design will be expanded to explore design space and capture important trends of new design and flow. Note that selection of parent designs, form of objective functions, and way to perform design morphing are also key techniques. Moreover, it is stated in [27] that high performance postprocess technique of a large CFD date is of importance as well, e.g., movie-style continuous presentation of geometry and flow between the two conceptually different ship designs provides novel intuitions of basic flow physics associated with the geometry, and those are found to be very useful



Fig. 15 Comparisons of limiting streamlines and axial vorticity distribution for 33CT. Full-scales condition (top and bottom for V and U types, respectively)

design information especially for young designers. Prognosis of extension of the present work in this respect involving "Big Data" research will be provided in future publications.

5 Example of application: duct design optimization

The next example is stern duct optimization test case, for which the geometrical configuration is shown in Fig. 17. This ESD is located near the stern and right before the propeller. The ship hull form is the 208,000 DW contractor design bulk carrier (referred to as 208BC), whose dimensions are: $L_{pp} \sim 295$ m, $B \sim 50$ m, $d \sim 16$ m, and design speed

 $V_{\rm s}$ ~14.5 kn. The aimed hydrodynamic effects are considered to be twofold: i.e., improvement of propeller inflow and generation of propulsion forces using the accelerated stern flow due to propeller suction effects. The real-life duct design must deal with the continuous variation of cross-sectional foil shapes, foil cord length, attack angle, foil thickness and location of installment, etc. Although all are possible to consider in the present optimization method, only longitudinal installment location and uniformly distributed attack angle are considered for the simplicity in the present demonstration. That is given by using a two-design variable optimization problem derived by the present VDA, which is formulated as follows:

Min.
$$\begin{cases} F_1 = t \\ F_2 = 1 - w_t \end{cases}$$
 (36)

Subject to
$$\alpha_i^l \le \alpha_i \le \alpha_i^u$$
 $(i = 1, ..., M),$ (37)

i.e., Eq. 31 and associated assumptions are used. Design parameters are duct open angle and longitudinal location; and explicit constrains are imposed on variation of design variables, i.e., $0^\circ \le \alpha_1 \le 20^\circ$ and $10\% D_{\rm P} \le \alpha_2 \le 30\% D_{\rm P}$ in order to avoid excessive geometrical features. Variables α_1 and α_2 are related to duct open angle and longitudinal location, respectively. Objective functions are evaluated by using NAGISA, with zero Fr model scale conditions ($Re \sim 6 \times 10^6$) and ship-point self-propulsion conditions are performed using a custom-design clockwise rotating propeller for this ship. Turbulence model is Modified Spalart-Allmaras (MSA) with system parameter cover = 20. The computational grids are for full-load and even-keel conditions and OO topology NAGISA usable with about 3 million cells for hull part (i.e., 353×81×105 in longitudinal, girth-wise, and radial directions, respectively, including both port and starboard sides) and 0.26 million cells for duct part (i.e., $57 \times 33 \times 145$). These setup conditions are found optimal for the present test case based on the aforementioned grid study and realistic turnaround. Meshing and overset preprocessing are performed by using AUTODES and GTOOL described earlier.

For the present test case, NMRI SBD with RC-MOGA module is used. Population number is 40 and UNDX system parameter $\alpha = 0.5$ and $\beta = 0.35$ (see [41]). Optimizations were performed up to 4 generations, and 160 new designs are automatically generated. Figure 18 shows distribution of the designs, where a set of Pareto optimal designs are indicated by blue squares. Note that the values in the figures are normalized by that of a representative design (denoted with red marker). Apparently, the trend of the objective functions is conflicting. The present approach successfully yields Pareto optimal set which involves 5 designs with significant differences in *t* and $1 - w_t$. The number of Pareto optimal



Fig. 16 Comparisons of pressure distributions and limiting streamlines for 33CT (left, middle, right for model scale RaNS, full scale RaNS, and potential flow results, respectively; top and bottom for V and U types, respectively)

design is considered to be enough for further evaluation, and the significant differences in t and $1 - w_t$ are related to the diversity of Pareto optimal designs. The diversity appears acceptable for the present setup with relatively narrow range of variable variation to avoid excessive geometry as mentioned earlier. In this investigation, we apply unit analysis, which consider that duct is part of propulsion system and propulsion factors are examined using resistance without duct case. The related formula is Eq. 31. Hence, resultant propulsion factors represent effects of power gain due to duct installment.

In the following, more details of design aspects of generated new designs are discussed in association with sensitivity analysis of design variables. As shown in Fig. 19, Pareto set designs (denoted by red and purple markers) are located in the range between 10° and 18° for open angle, and 13 and 24%D for longitudinal location. In the comparison of those variables vs. resistance Cd (see Fig. 20), minimum is nearly located at open angle 10° , while longitudinal location does not indicate clear minima or maxima, all of which agrees well with our design and EFD experiences.

Next, influences of design variables on propulsion factors are discussed. As is shown in Fig. 21, minimum thrust deduction t is indicated near open angle 13°, which is attributed to a fact that duct inflow angle is larger near the top of duct due to propeller suction effects. On the other hand, open angle around 5° yields highest wake coefficient $1 - w_t$, where the angle seems to be adjusted angle with the inflow that does not cause deceleration of propeller inflow. Again, those trends indicated in the results agree well with our design experiences.

Next, flow field aspects are discussed. Figure 22 shows comparison of surface pressure field and flow separation region (iso-surface of U=0) for id051 design and its openangle variated versions. id051 is one of Pareto set designs and finally selected as EFD validation. Open angles 0° and 20° are considered to show the influences on flow. For all cases, flow separation near the stern shown in towing condition generally decreases in self-propulsion condition due to the accelerated flow by propeller suction effects; where the separation region is smallest for id051 in self-propulsion condition. The flow separation for angle 0° and 20° occurs in upper and lower region of the duct, respectively. The smallest separation region for id051 is related to the smallest t among the three, and it is also seen that thrust is generated especially upper part of the duct.

On the other hand, the trends of flow shown in the Pareto optimal designs are noteworthy. Table 3 shows the comparison of design variables among Pareto optimal designs in the descending order of t, and the Fig. 23 the respective surface

274



Fig. 17 Overview of stern geometry and installment of stern duct (208BC duct optimization test case)



Fig. 18 Distribution of new designs from the present MOGA (208BC duct optimization test case)

pressure fields for the self-propulsion condition. As shown in the table and figure, differences in those designs are mainly attributed to differences in pressure field around the duct, which had direct influences on the acceleration and deceleration of flow in the region. Through those considerations in association with discussions for Eq. 31 and the impact of the terms, id051 is selected for EFD validation.

Table 4 shows the comparison of CFD and EFD results for the selected design, where the gain due to ESD is clearly



Fig. 19 Distribution of solutions with respect to design variables in MOGA optimization (208BC duct optimization test case)

seen in both results and, although the magnitudes are over and under predicted within allowable error, the trends shown in EFD are also correctly predicted by the present CFD. Our investigation suggests that trends in $\Delta(1-t)$ and $\Delta(1-w_t)$ are mostly related to flow aspect around upper and lower half of duct, respectively. The duct thrust, which has a large influence on (1 - t), is mostly generated in the upper half and flow separation must be avoided. Gain in $(1 - w_t)$ is attributed to decelerated flow in the region near the lower half of duct. Finally, estimated ESD power gain in consideration of tank test is 4.6% for Fr = 0.140, which is mainly attributed to gain in (1 - t) which is suggested in Eq. 30, i.e., contribution of (1 - t) is nearly twice as large as that of $(1 - w_t)$. Those trends predicted by CFD agree well with our design experience and the present optimization of duct design is overall successful in our judgement.

6 Example of application: evaluation of energy saving effects of pre-swirl stern-fin

Evaluate
$$\begin{cases} F_1 = 1 - t \\ F_2 = 1 - w_t \end{cases}$$
 (38)

The next example is pre-swirl stern-fin (PSF) test case, where the present VDA is used to evaluate hydrodynamic effect of ESD. Target functions are defined as per Eq. 38. This ESD is also located near the stern and right before the propeller, and the same energy saving effects as those for the previous stern duct ESD are aimed. That is, improvement of propeller inflow and generation of propulsion forces using the accelerated stern flow due to propeller suction effects. The ship hull form is a 21,000 DW contractor design tanker (referred to as 21T), whose dimensions are: $L_{wl} \sim 144$ m, $B \sim 24$ m, $d \sim 10$ m, and design speed $V_s \sim 14.5$ kn. EFD

0.010

3



0.010

Fig. 21 Influence of design variables on propulsion factors (208BC duct optimization test case)

Fig. 20 Influence of design

variables on resistance (208BC duct optimization test case)

and CFD are performed for the baseline design of hull and ESD, to identify important flow features to be considered in design optimization. CFD method is NAGISA along with zero Fr model scale conditions ($Re \sim 6 \times 10^6$) and ship-point self-propulsion conditions are performed using a customdesign clockwise rotating propeller for this ship. Turbulence model is Modified Spalart-Allmaras with system parameter cover = 20. The full-load and even-keel conditions are assumed. Topology and total number of computational grids are OO type and about 4.6 million cells, where about 3 million cells for hull part and 1.6 million cells for ESD and rudder. As is the previous case, meshing and overset preprocessing are performed using AUTODES and GTOOL, and the acceptable accuracy.

Figure 24 shows the CFD results for surface pressure field and flow separation region (iso-surface of U=0) for baseline design. It is seen that for the latter separation region is reduced due to acceleration of flow by propeller suction effects, and stern surface pressure is generally decreased. Rudder surface pressure indicates similar trend, i.e., decrease of pressure in the mid-body region due to accelerated flow, but near the leading edge where the stagnation pressure increases and the peak areas are clearly shifted in different direction in the upper and lower regions of rudder, which is due to the rotated flow right after propeller.

Fig. 22 Comparison of surface pressure field and flow separation region (iso-surface of U=0) for id051 design and its open-angle variated versions. id051 is one of Pareto set designs. Left and right correspond to towing and selfpropulsion conditions, respectively (208BC duct optimization test case)



id051 (Open Angle 13.2 deg.)





Those overall flow features agree with our design intuition, and the present CFD appears to accurately capture those. Table 5 shows comparison of CFD and EFD results of target functions, i.e., self-propulsion factors, where the trends shown in EFD are correctly predicted by the present CFD within allowable error in our judgment. It is noteworthy in detailed flow aspects shown for each fin, i.e., Figs. 25 and 26 compare detailed flow aspects around each fin for towing and self-propulsion conditions. It is noteworthy that each fin plays its own role to effectively perform as ESD, i.e., starboard side fins for better propeller inflow arrowing relatively large flow separation, in contrast, port side fins for better trust effects avoiding flow separation. This information is

id	Open angle (°)	Position (% D_p)	
id051	13.2	13.3	
id023	11.1	14.4	
id040	13.8	23.4	
id046	16.7	16.6	
id047	16.4	16.5	

 Table 3
 Comparison of design variables for Pareto optimal designs

 (208BC duct optimization test case)

considerably useful to optimize fin arrangement along with fin shape in further investigation.

19.8

7 Concluding remarks

17.3

In this paper, for the first time, basic idea and examples of application of VDA were proposed and presented. Main aim of the method is to overcome the issues claimed for current CFD with limited level of accuracy especially for prediction of power, i.e., DP or EP. Instead of just counting on predicted value and solve a SOOP to minimize, we first introduce VDA to decompose a target ship performance function into terms including embedded parameters, then formulate and solve a MOOP. Any scheme to solve MOOP can be applied. An overview of the present approach was given and results were discussed including comparison with available EFD data, and detailed analysis of flow and integral parameters were presented. The effectiveness of the present approach was also discussed.

First, detailed evaluation of the present approach associated with practical design projects was discussed, involving exploration of its design space and investigation of hull form and associated flow features. Test parent design is two types: chemical tanker and bulk currier hull forms designed in NMRI EEDI Project Team. It is shown that the present SBD in association with VDA of target function effectively cooperates with the traditional design approach. The hull form blending is shown to be a very promising approach, while selection of parent designs and form of objective functions are also key techniques. Moreover, as stated in [27], high performance post-process technique of a large CFD date is of importance as well, e.g., movie-style continuous presentation of geometry and flow between the two conceptually different ship designs provides novel intuitions of basic flow physics associated with the geometry, and those are found to be very useful design information especially for young designers.

The next example was stern duct optimization test case, for which the shape of ESD located near the stern and right before the propeller is optimized. The ship hull form is the

🖄 Springer



Fig. 23 Comparison of surface pressure field for Pareto set designs. For self-propulsion conditions. (208BC duct optimization test case)

id139

 Table 4 Comparison of results between CFD and EFD for the selected Pareto optimal design, id051 (208BC duct optimization test case)

	EFD	CFD	ΔE/(EFD w/o WAD) (%)
$\Delta(1-t)$	0.021	0.015	-0.70
$\Delta(1-w_t)$	-0.023	-0.005	2.70
$\Delta \eta_{\rm R}$	-0.003	-0.002	0.10

 Table 5
 CFD and EFD results of self-propulsion factors for the baseline design (21T PSF test case)

	EFD	CFD	
$\frac{\Delta(1-t)/(1-t)}{2}$	-0.110	-0.080	
$\frac{\Delta(1-w_t)/(1-w_t)}{2}$	-0.044	-0.036	

 $\Delta V/V = (V_2 - V_1)/V_1$, where V_1 and V_2 correspond to V for without and with fins, respectively

contractor design bulk carrier. This test case was also successfully demonstrated, and the trends shown in the results between the baseline and optimal designs agree well with those of EFD. Detailed discussion regarding geometry and flow were given, and again the validity of the present approach was indicated. The third and final examples are related to PSF test cases where fins are attached near the stern end and right before the propeller. The results are again seen very favourable. The present VDA can be used to evaluate energy saving effect of ESD, i.e., predicted results show good agreement with EFD data. The results indicate noteworthy fact that each fin plays its own role to maximize ESD performance. Optimal PSF design will be extremely difficult without SBD tools like one demonstrated in this study.

In conclusion, based on all of achievements discussed above, the present approach is shown very promising and worthy for further evaluation and applications. Design optimization as well as the exploration of design space demonstrated in the present work was not be efficiently possible without utilizing the present approach, i.e., SBD with detailed observation of decomposed target parameters. The propulsive factors in this case are theoretically derived by the present VDA. Solution method based on MOOP scheme was shown capable. Since the scheme is effective to handle with different accuracy objective functions but with trustworthy trends. Once MOOP is introduced in optimization framework, consideration of other design functions, e.g., seakeeping merit functions or structural merit functions, is possible in a straightforward manner. Now, we have a confidence that, if our approach is properly used, the knowledge and capability of the traditional design will be expanded to explore design space and capture important trends of new design and flow.



Fig. 24 Comparison of computed surface pressure field and flow separation region (iso-surface of U=0) for baseline design. Top and bottom (**a**, **b**) correspond to towing and self-propulsion conditions, respectively (21T pre-swirl stern-fin (PSF) test case)

(a) **(b)** (c) (**d**) 0.45 (e)

Fig. 25 Fins **a**–**e** local view of computed surface pressure field and flow separation region (isosurface of U=0) for baseline design (21T PSF test case for towing condition)

280

Fig. 26 Fins **a**–**e** local view of computed surface pressure field and flow separation region (isosurface of U=0) for baseline design (21T PSF test case for self-propulsion condition)



D Springer

Acknowledgements This work has been supported by Grant-in-Aid for Scientific Research, Japan (Project nos. 24360363 and 15H04217). The authors would like to express their appreciation to those who concern for the support and encouragement. The authors' appreciation is extended to Mr. Kenichi Kume (NMRI) for extensive CFD data (duct series self-propulsion simulation by using unstructured meshing tool and CFD); and Mr. Yoshihisa Okada and Mr. Kenta Katayama (Nakashima Propeller Co., LTD) for valuable advice on duct optimization problem.

References

- Proceedings 25th international towing tank conference, Fukuoka, Japan, 14–20 September, 2008
- Proceedings 26th international towing tank conference, Rio de Janeiro, Brazil, 28 August–3 September, 2011
- Proceedings 27th international towing tank conference, Copenhagen, Denmark, 31 August–5 September, 2014
- Hart CG, Vlahopoulos N (2010) An integrated multidisciplinary particle swarm optimization approach to conceptual ship design. Struct Multidiscip Optim 41:481–494
- Kuhn J, Collette M, Lin W-M, Schlageter E, Whipple D, Wyatt D (2010) Investigation of competition between objectives in multiobjective optimization. Proc 28th Symp Nav Hydrodyn 2:1233–1244
- Zhang B-J, Ma K (2011) Study on hull form optimization for minimum resistance based on niche genetic algorithms. J Ship Prod Des 27:162–168
- He XD, Hong Y, Wang RG (2012) Hydroelastic optimization of a composite marine propeller in a non-uniform wake. Ocean Eng J 39:14–23
- Hollenbach U, Reinholz O (2011) Hydrodynamics trends in optimization propulsion. In: Second international symposium on marine propulsors SMP'11, Hamburg, Germany
- Stuck A, Kroger J, Rung T (2011) Adjoint-ased Hull Desogn for wake optimization. Ship Technol Res 58(1):34–44
- Luo W, Fu B, Guedes Soares C, Zou Z (2013) Robust control for ship course-keeping based on support vector machines, particle swarm optimization and L2-Gain. OMAE 2013, Nantes
- Xu Y, Sun Y, Wei Y, Guan H, Liu M, Cai W (2012) Study on ship–ship hydrodynamic interaction by ANN optimization. MAR-SIM 2012, Singapore
- Kim HJ, Choi JE, Chun HH (2016) Hull-form optimization using parametric modification functions and particle swarm optimization. J Mari Sci Technol 21(1):129–144
- Chrismianto D, Kim DJ (2014) Parametric bulbous bow design using the cubic Bezier curve and curve-plane intersection method for the minimization of ship resistance in CFD. J Mar Sci Technol 19(4):479–492
- 14. Diez M, He W, Campana EF, Stern F (2014) Uncertainty quantification of Delft catamaran resistance, sinkage, and trim for variable Froude number and geometry using metamodels, quadrature and Karhunen–Loeve expansion. J Mar Sci Technol 19(2):143–169
- 15. Han S, Lee YS, Choi YB (2012) Hydrodynamic hull form optimization using parametric models. J Mar Sci Technol 17(1):1–17
- Kandasamy M, Peri D, Ooi SK, Carrica P, Stern F, Campana EF, Osborne P, Cote J, Macdonald N (2011) Multi-fidelity optimization of a high-speed foil-assisted semi-planing catamaran for low wake. J Mar Sci Technol 16(2):143–156

- Tahara Y, Tohyama S, Katsui T (2006) CFD-based multi-objective optimization method for ship design. Int J Numer Methods Fluids 52:449–527
- Tahara Y, Peri D, Campana EF, Stern F (2008) Computational fluid dynamics-based multiobjective optimization of a surface combatant. J Mar Sci Technol 13(2):95–116
- Tahara Y, Peri D, Campana EF, Stern F (2011) Single and multiobjective design optimization of a fast multihull ship: numerical and experimental results. J Mar Sci Technol 16(4):412–433
- Campana EF, Peri D, Tahara Y, Stern F (2006) Shape optimization in ship hydrodynamics using computational fluid dynamics. Comput Methods Appl Mech Eng 196:634–651
- Kandasamy M, Peri D, Tahara Y, Wilson W, Miozzi M, Georgiev S, Milanov E, Campana EF, Stern F (2013) Simulation based design optimization of waterjet propelled Delft catamaran. ISP 60:277–308
- 22. Diez M, Campana EF, Stern F (2015) Design-space dimensionality reduction in shape optimization by Karhunen–Loeve expansion. Comput Methods Appl Mech Eng 283:1525–1544
- Kim K, Leer-Andersen M, Orych M (2014) Hydrodynamic optimization of energy saving devices in full scale. In: 30th symposium on naval hydrodynamics, Hobart, Tasmania, Australia, 2–7 November, (CDROM)
- Gothenburg 2010 (2010) A workshop on numerical ship hydrodynamics, Gothenburg, Sweden, 8–10 December, (CDROM)
- Tokyo 2015 (2015) A workshop on numerical ship hydrodynamics. 2–4 December, (CDROM), Tokyo
- Kasahara Y (2015) Hull form design utilizing CFD for improvement of EEDI. In: International workshop on ship technologies related to energy efficiency design index (EEDI), Tokyo, Japan
- Ichinose Y, Tahara Y, Kasahara Y (2015) Numerical study on flow field around the Aft part of hull form series in a steady flow. In: 18th numerical towing tank symposium, Cortona, Italy, (CDROM)
- NMRI CFD 2015 (2016) User manual and related information. National Maritime Research Institute, Mitaka
- HEXPRESS (2016) User manual and related information. http:// www.numeca.com/
- Ichinose Y, Kume K, Tahara Y (2016) A development and analysis of the new energy saving device "USTD". In: Proceedings of the 19th numerical towing tank symposium (NuTTs'16), St. Pierre d'Oleron, France, 3–4 October, (CDROM)
- Tank Test Report (2015) Tank Test No. 15-04, (Japanese, unpublished). National Maritime Research Institute, Mitaka
- Tahara Y, Saitoh Y, Matsuyama H, Himeno Y (1999) CFD-aided optimization of tanker stern form—2nd report: minimization of delivered horse power. J Kansai Soc Nav Archit Jpn 232:9–18
- Breslin JP, Andersen P (1994) Hydrodynamics of ship propellers. Cambridge University Press, Cambridge, p 194
- Spalart PR, Allmaras SR (1994) An one-equation turbulence model for aerodynamic flows. La Recherch Aerospatiale, No. 1
- 35. Wilcox DC (1994) Simulation of transition with a two-equation turbulence model. AIAA J 32(2):247–255
- NAPA (2016) User manual and related information. http://www. napa.fi/
- RHINOCEROS (2016) User manual and related information. http://www.rhino3d.com/
- GRIDGEN (2016) User manual and related information. http:// www.pointwise.com/
- Tahara Y (2015) CFD-based hull form/appendage optimization by using deterministic and stochastic optimization theory. In: 12th international marine design conference, Tokyo, Japan, 11–14 May 2015 (CDROM)

- 40. Tahara Y, Shingo S, Kanai A (2017) CFD based optimal design method for energy saving devices by using overset grid technique and nonlinear optimization theory. J Jpn Soc Nav Archit Ocean Eng 26:1–16
- 41. Deb K (2001) Multi-objective optimization using evolutionary algorithms. Wiley, Hoboken
- 42. Hess JL, Smith AMO (1962) Calculation of non-lifting potential flow about arbitrary three-dimensional bodies. Douglas Report No. E S 40622. Douglas Aircraft Company, Santa Monica