11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

NUMERICAL ANALYSIS OF TURBULENT FLOW AROUND ENERGY SAVING PRE-SWIRL STATOR FOR FULL AND MODEL SCALE SHIPS

Sunho PARK[1], Gwangho OH[2], Shin Hyung RHEE[3*], Bong-Yong KOO[4] and Hoseong LEE[4]

[1] Department of Ocean Engineering
Korea Maritime and Ocean University
606-791, 727 Taejong-ro, Yeongdo-gu Busan, Korea
Email: spark@kmou.ac.kr, oe.kmou.ac.kr

[2] NextFOAM Co., LTD.
153-790, Seoul, Korea
Email: gwangeo@nextfoam.co.kr, www.nextfoam.co.kr

[3] Department of Naval Architecture and Ocean Engineering, RIMSE
Seoul National University
151-742, 1 Gwanak-ro, Gwanak-gu, Seoul, Korea
Email: shr@snu.ac.kr, snutt.snu.ac.kr

[4] American Bureau of Shipping
Jungang-dong 4-Ga, Jung-gu, Busan, Korea,
Email: bkoo@eagle.co.kr, hlee@eagle.co.kr, www.eagle.org

Key Words: *full Scale Performance Predictions, Energy Saving Device (ESD), Pre-swirl Stator (PSS)*

Abstract. Recently interest on energy saving devices (ESD's) has increased with the enforcement of the energy efficiency design index (EEDI) verification proposed by the international maritime organization (IMO). Extension of propulsive performance results from model to full scale ships plays an important role in verification of ships with ESD's. The present study proposed a reliable and efficient propulsive performance prediction method for full scale ships with ESD's. The propulsive performance prediction in full scale KVLCC2 with pre-swirl stator (PSS) was conducted by the proposed method. Its results were then compared with those by the existing extension methods and by full scale CFD computations. From the results, it was confirmed that the proposed method could extend the model scale results to full scale ones with ESD's performance improvement effects. Unlike the existing methods, it takes into account of ESD's and consumes much less computational resources and time than full scale CFD computations.

1 INTRODUCTION

IMO proposed EEDI with the demand for improved fuel efficiency, low emissions and optimized operation. To attain required EEDI, enhancement of the propulsion performance is one of the easiest and most efficient methods. Thus, interest on ESD's has increased recently.

A large amount of researches on ESD's has been done experimentally and computationally over the past decade. Korkut[1] carried out resistance and self-propulsion experiments for a partial wake equalizing duct. Celik [2] studied an effect of a wake equalizing duct on the propulsion performance using CFD and showed 10% maximum power gain. Celik and Guner [3] studied PSS using lifting line theory and CFD. Hansen et al. [4] evaluated the efficiency of propeller boss cap fin (PBCF) in the model and sea trial tests. Kawamura et al. [5] investigated the influence of the Reynolds number and inflow condition on PBCF performance using CFD. Lee et al. [6] evaluated the propulsion performance under calm water and seaway conditions for energy saving rudder fin using CFD by the concept of energy-loss analyses. Choi et al. [7] developed an ESD and compared quasi-propulsive efficiencies predicted by modified ITTC78 and ITTC 99 methods. Shin et al. [8] studied pre-swirl duct for self-propulsion and cavitation performances numerically and experimentally. Hollenbach and Reinholze [9] tested many ESDs, such as propulsion improving device, PSS, thrust fin, stator fin, and showed power gains by model tests. ESDs decreased the delivered power by 3% to 6%. In any event, however, many studies were restricted to the model scale and focused on flows around ESD's. To verify EEDI for ships with ESD's, extension of propulsive performance results from model to full scale ships is necessary, because EEDI verification is required by classification companies for all newly built ships in operating conditions.

In the present study, the objectives were therefore (1) to propose a reliable and efficient propulsive performance prediction method for full scale ships with ESDs and (2) to validate the proposed method.

The paper is organized as follows. The description of full scale prediction methods is presented first, and followed by the physical model problem and computational methods for validation. The results are then presented and discussed. Finally, a summary and conclusions are provided.

2 FULL SCALE PERFORMANCE PREDICTION METHODS FOR ESD'S

2.1 Review of existing methods

To predict the propulsive performance of full scale ships, the ITTC 1978 method is commonly used. The flow chart of the ITTC 1978 method is shown in Figure 1. Here, w_t, w_v, and w_{inv} indicate the total wake fraction, viscous and inviscid components of the total wake fraction, respectively. The subscripts 'w/ ESD' and 'w/o ESD' indicate the case with and without ESD, respectively. The subscripts 'model' and 'full' indicate the model- and full scale ship, respectively. C_A is the influence of hull roughness to be considered in the full scale. Here, C_A of 0 is assumed. w_t could be decomposed into w_v and w_{inv}. w_{inv} is independent of the Reynolds number, thus $w_{inv,\, w/ ESD,\, full}$ must be the same as $w_{inv,\, w/ ESD,\, model}$. w_v is dependent of the Reynolds number, thus $w_{v,\, w/\, ESD,\, full}$ needs to be calculated from $w_{v,\, w/\, ESD,\, model}$ by multiplying the ratio of the viscous drag of the model to full scale ships. w_{inv} is assumed to be the sum of the thrust deduction factor ($t_{w/\,ESD,\,model}$) and a constant value of 0.04 in the ITTC 1978 method. The constant value is due to the rudder effect, an empirical value. The ITTC 1978 method is proved to work properly for hulls without appendages. However, it is not designed and validated for ships with ESD's.

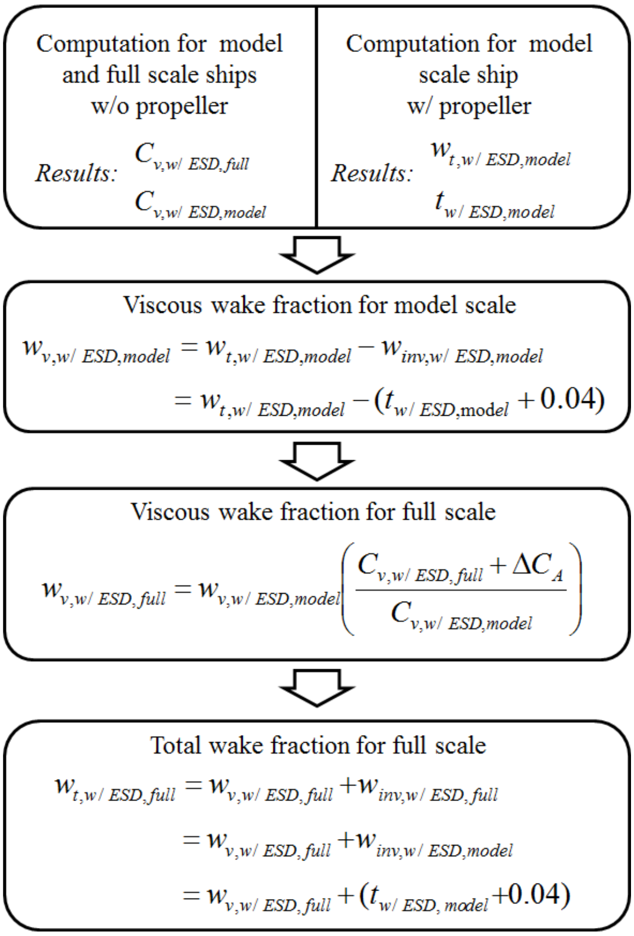

Figure 1: Flow chart of ITTC 1978 method

To consider ESD's, the ITTC 1999 method was proposed. The principal concept of the ITTC 1999 is that the ratio of the model to full scale ship without ESD's must be the same as that with ESD's. $w_{t,\ w/\ ESD,\ full}$ is calculated using $w_{t,\ w/o\ ESD,\ full}$, $w_{t,\ w/\ ESD,\ model}$, and $w_{t,\ w/o\ ESD,\ model}$. The flow chart of the ITTC 1999 method is shown in Figure 2. To apply this method, computations for the model and full scale ships without ESD's, and the model scale ship with ESD's should be carried out. Obviously, it requires huge computational resources and time for the full scale computation.

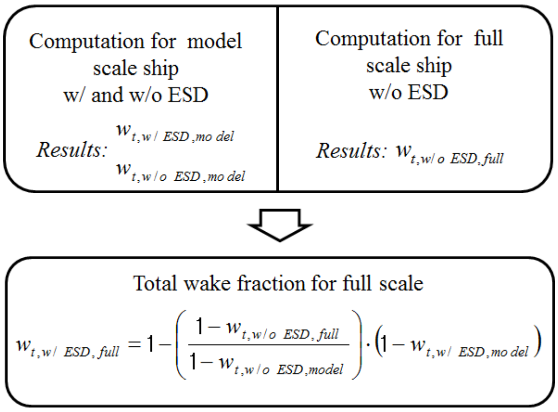

Figure 2: Flow chart of ITTC 1999 method

Full scale CFD computations for ships with ESD's are also possible to predict the propulsive performance without using extension methods. However, the credibility of full scale computations is still being disputable, ye the required computational resources and time is a huge burden.

2.2 Proposed method

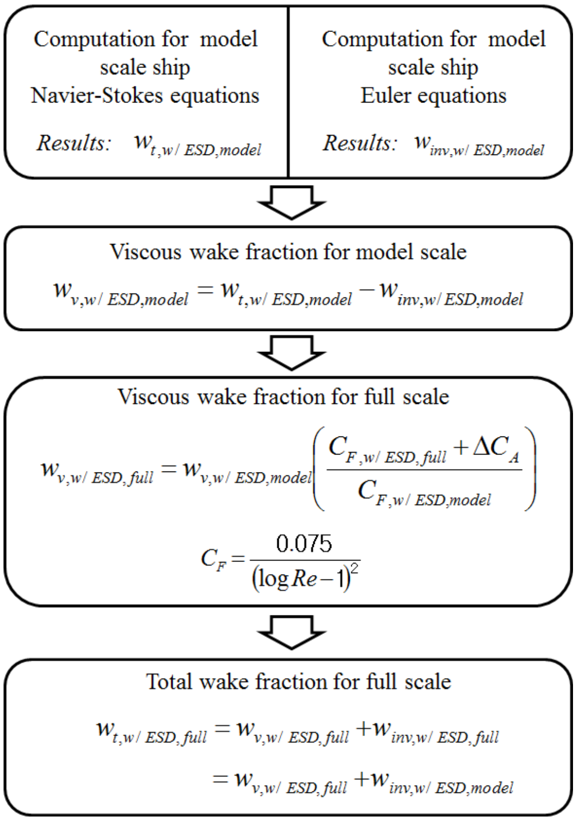

Figure 3: Flow chart of proposed method

To overcome the shortcomings of the existing methods, a simple extension method with ESD's efficacy in mind is proposed. In the ITTC 1978 method, w_{inv} is calculated empirically. However, in the proposed method, w_{inv} is accurately calculated from the solution of the Euler equations. In other words, to get the inviscid component, the Euler equations are solved numerically for a ship with ESD. In this way, w_{inv} can take into account of the performance improvement by an ESD. The ratio of $w_{v,\, w/\,ESD,\, model}$ to $w_{v,\, w/\,ESD,\, full}$ is calculated by the ratio of the model to full scale friction coefficients, which was adopted by the ITTC 1957, instead of the computational results for the full scale ships, which required huge computational resources and time. In summary, the newly proposed method estimates w_{inv} more precisely by utilizing the solution of the Euler equations, and takes into account of the Reynolds number effect by the ITTC 1957 friction formula, which is simple, yet proved reliable. The flow chart for the proposed prediction method is shown in Figure 3.

3 MODEL PROBLEM

PSS was selected as the ESD of interest for the present study. The detailed shape of PSS was described in the Korea patent registered by DSME [10]. PSS, mounted in front of the propeller, has three stator blades on the port side and one stator blade on the starboard side. The straight upward position was 0^{o} and the angle increased with clockwise direction looking from downstream. The blades were located at 90^{o}, 225^{o}, 270^{o}, and 315^{o}. The left blades have pitch angle of 17^{o}, 19^{o}, 23^{o} from top to bottom, respectively. The right blade has pitch angle of 22^{o}. Figure 4 shows PSS, propeller and rudder looking from downstream.

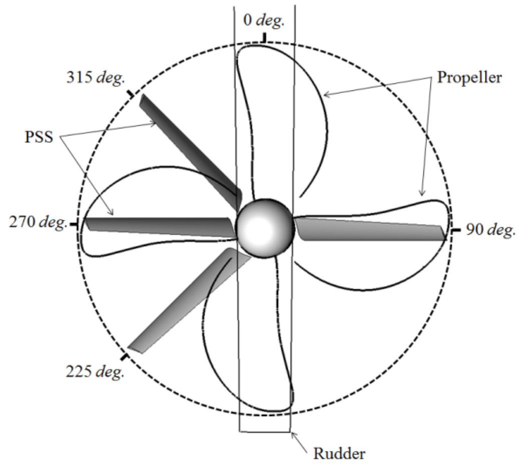

Figure 4: Pre-swirl stator, propeller and rudder

Hollenbach and Reinholz [9] tested many ESD's in the HSVA test facility and showed power gains. When PSS was installed, power gains for tankers were relatively larger than those for container ships. To identify the effect more clearly, the KRISO very large crude oil carrier 2 (KVLCC2) was selected as the object ship for the present study. KVLCC2 was designed as representative of full form ships by the Korean Research Institute of Ships and Ocean Engineering (KRISO). Principle particulars of KVLCC2 are listed in Table 1. Test conditions are summarized in Table 2.

Table 1: Principal particulars of KVLCC2.

	Full scale ship	Model scale ship
Scale	1	1/100
Lpp (*m*)	320	3.2
B (*m*)	58	0.58
T (*m*)	20.8	0.208
Displacement (m^3)	312,622	0.3126
Wetted surface area (m^2)	27,467.3	2.7467
Propeller diameter (*m*)	9.86	0.0986

Table 2: Test conditions.

	Full scale ship	Model scale ship
U_∞ *(m/s)*	0.797	7.974
Re	2.5×10^6	2.5×10^9
Fr	0.142	

4 COMPUTATIONAL METHODS

A pressure-based cell-centered finite volume method was employed along with a linear reconstruction scheme that allows the use of computational cells of arbitrary shapes. The solution gradients at the cell centers were evaluated by the least-square method. The convection terms were discretized using the second-order accurate scheme, and for the diffusion terms, a central differencing scheme was used. For turbulence closure of the Reynolds-averaged Navier-Stokes (RANS) equations, the realizable k-ε turbulence model [11] was adopted. The wall function was used for the near-wall treatment [12]. The velocity-pressure coupling and overall solution procedure were based on a semi-implicit method for pressure linked equations (SIMPLE) type segregated algorithm adapted to an unstructured grid. The discretized algebraic equations were solved using a point-wise Gauss-Seidel iterative algorithm, while an algebraic multi-grid method was employed to accelerate solution convergence. STAR-CCM+, commercially available CFD code, was used for mesh generations and computations.

5 RESULTS AND DISCUSSION

5.1 CFD computations

All the full scale propulsive performance predictions by the full scale computations, ITTC 1978, ITTC 1999, and proposed methods were based on CFD computations. Thus, the accurate CFD computations for the model and full scale ships were crucial for the present study.

In the Cartesian coordinate system adopted here, the positive x-axis was in the streamwise direction, the positive y-axis was in the starboard direction, and the positive z-axis was in the upward direction. The solution domain extent was $-2 \le x/l \le 3$, $-1.5 \le y/l \le 1.5$, and $-1 \le z/l \le$

0. Here, l indicates the length of the ship. The upstream inlet and far-field boundary was specified as the Dirichlet boundary condition, i.e., with a fixed value of the velocity. On the downstream exit boundary, the reference pressure with the extrapolated velocity was applied. The free-surface wave was ignored, i.e. a double body model with a rigid and flat free-surface was considered, and thus slip condition was applied on the top boundary. No-slip condition was applied on the ship surface. To simulate the propeller revolution, the generic grid interface (GGI) method was selected. The time step of 0.001 second was used.

The mesh generation was also carried out using STAR-CCM+. The cut-cell method was used to generate volume meshes. A single-block mesh of 3.8 million hexahedral cells for KVLCC2 with PSS and of two million hexahedral cells for KVLCC2 without PSS was used.

Because PSS was located in front of the propeller, accurate reproduction of the wake flow was critical for successful analysis of the flow around PSS. Figure 5 shows nominal wake and cross-flow (*yz*-plane) velocity vectors on the propeller plane of the model scale ship measured by Seo et al. [13]. Figure 6 shows computed results for the same. The so-called hook-shaped vortex flow was well captured and the upward flow on the side and inward turning flow on the top were clearly reproduced.

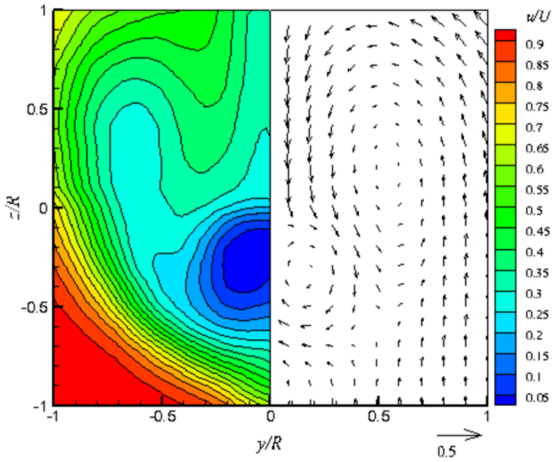

Figure 5: Nominal wake measured by Seo et al. [13].

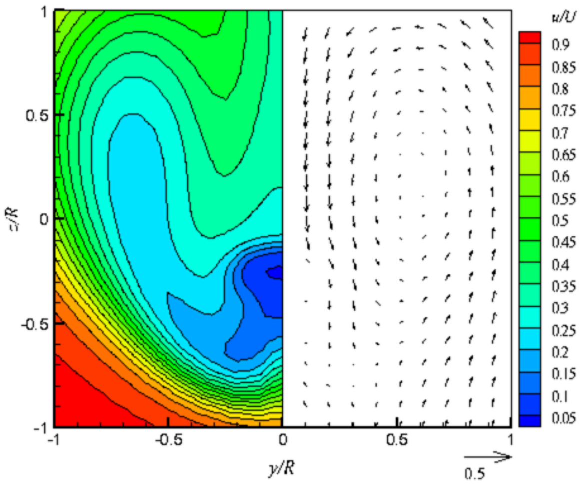

Figure 6: Computed nominal wake.

The computation was carried out for 20 seconds. In the computations of the model scale ship, the self-propulsion point was identified when the resistance on the hull and rudder (R) was same as the sum of the towing force (F_D) and the thrust of the propeller (T). The self-propulsion points for KVLCC2 with and without PSS were 517 *rpm*, and 519 *rpm*, respectively. The revolution rate decreased due to PSS. For full scale ships, the self-propulsion point was identical when the resistance on the hull and rudder (R) was same as the thrust of the propeller (T). For the full scale ship, the towing force was not considered. The self-propulsion points for KVLCC2 with and without PSS in the full scale ship were 68.4 *rpm* and 71.5 *rpm*, respectively. The revolution speed difference increased in the full scale, indicating that PSS was more effective for the full scale ship. Kawamura et al. [5] showed better efficacy of PBCF in the full scale than that in the model scale. The influence of PSS was greater because the boundary layer thickness nondimensionalized by L_{PP} (δ/L_{PP}) was smaller in high Reynolds number.

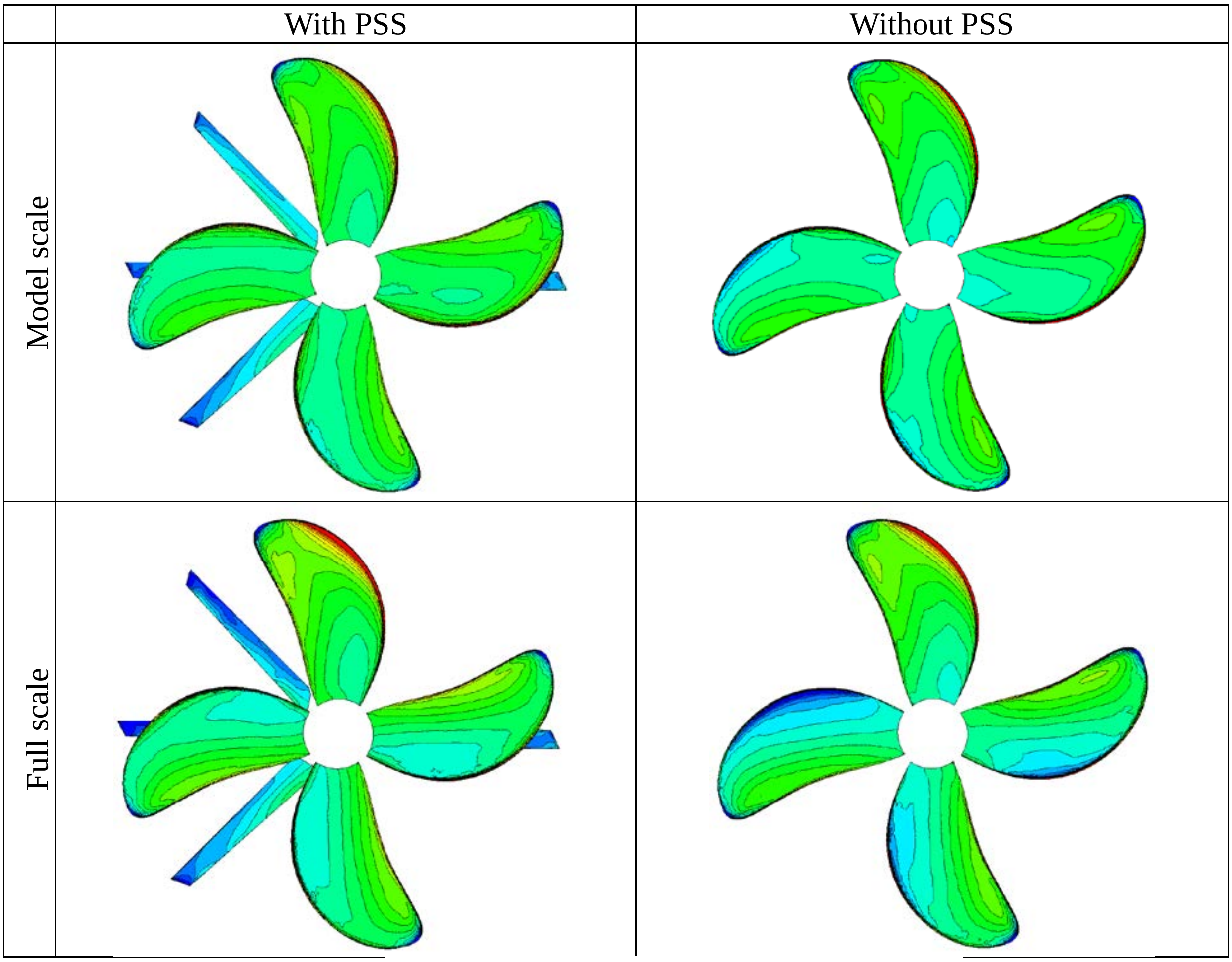

Figure 7: Pressure coefficient contours on pressure side of blade

Figures 7 and 8 show the pressure coefficient contours on the face and back of the propeller, respectively, when the top propeller blade is located at 0^o, which is defined in Figure 4. The pressure coefficient ($C_P = (P - P_{ref})/0.5\rho U_\infty^2$) was based on the reference pressure (P_{ref}), fluid density (ρ), and freestream velocity (U_∞). The influence of PSS, displayed by C_P difference, was more prominent on the blades at 90^o and 270^o, because they were located right behind the PSS fins. Compared to the model scale, it was observed that the change in C_P is greater in the full scale. The thinner boundary layer and wake must have played a significant role here. The efficacy of PSS was confirmed by the increased amount of pressure difference between face and back of the propellers, when PSS was installed.

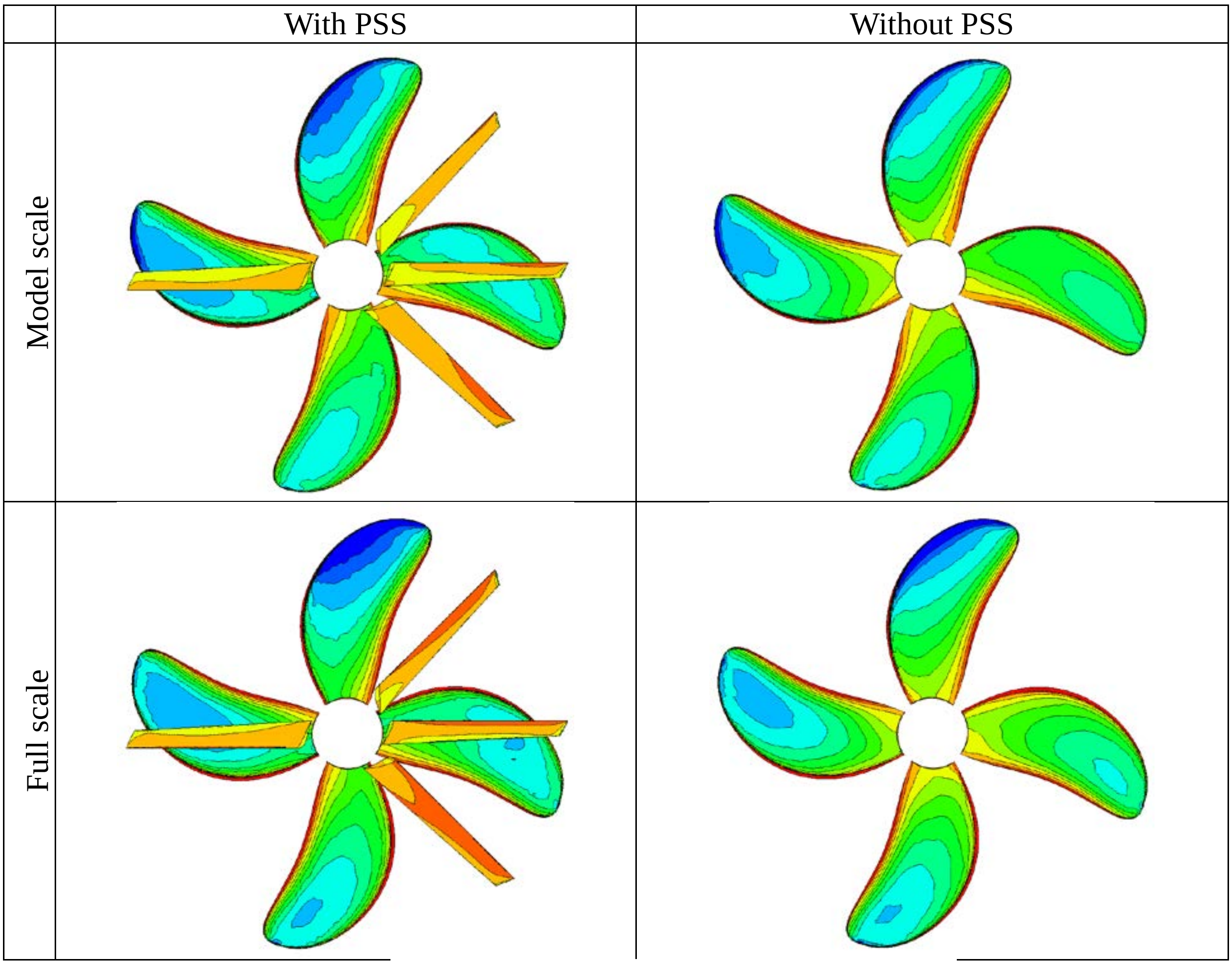

Figure 8: Pressure coefficient contours on suction side of blade

The minimum pressure level is closedly related to cavitation. Cavitation takes place when the local pressure is lower than the vapor pressure. In other words, cavitation occurs when $-C_P$ is equal to or less than the cavitation number ($\sigma = (P_{ref} - P_v)/0.5\rho U_\infty^2$) of the flow. The minimum -C_P for the blade at 0^o with and without PSS for the model scale was 4.4 and 4.5, respectively.

For the full scale, the minimum $-C_P$ for the blade at 0^{o} with and without PSS was 7.2 and 7.6, respectively. It is of noteworthy that, in terms of cavitation inception, PSS might have negative effects, when it is not carefully designed.

Figure 9 shows the nondimensionalized x-velocity (u/U_∞) contours and cross-flow velocity vectors on the wake plane at x/L_{PP} of 0.0226, which was a mid-point between the propeller and PSS, when the top propeller blade was located at 0^{o}. Here, the center of the rudder stock was at x/L_{PP} of 0. The cross flow velocity vectors without PSS were similar to those of nominal wake. Figure 10 shows the direction of the propeller rotation and the cross-flow patterns. The right-handed propeller rotates clock-wise direction looking from downstream. On the starboard side, the directions of propeller rotation and the cross-flow were opposite, while, on the port side, the directions were congruent. PSS basically modifies the flow in to the propeller blades in a favorable way. It increases the angle of attack into the blades, especially on the port side. Although PSS produces small scale vortices and makes the x-velocity low, they are relatively minor side effects.

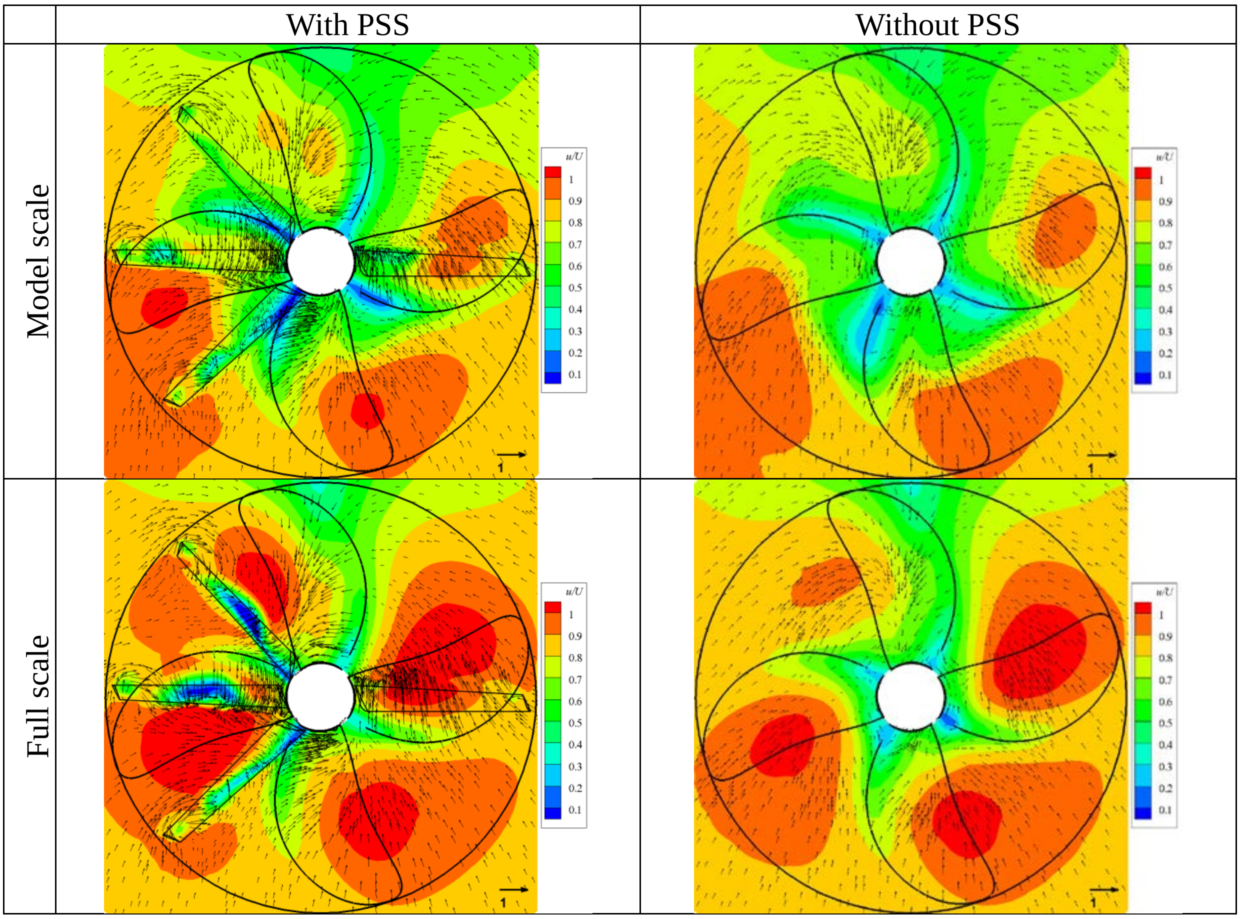

Figure 9: Nondimensionalized x-velocity contours and cross-flow velocity vectors at *x/Lpp*=0.0226 with the blade of 0^{o}

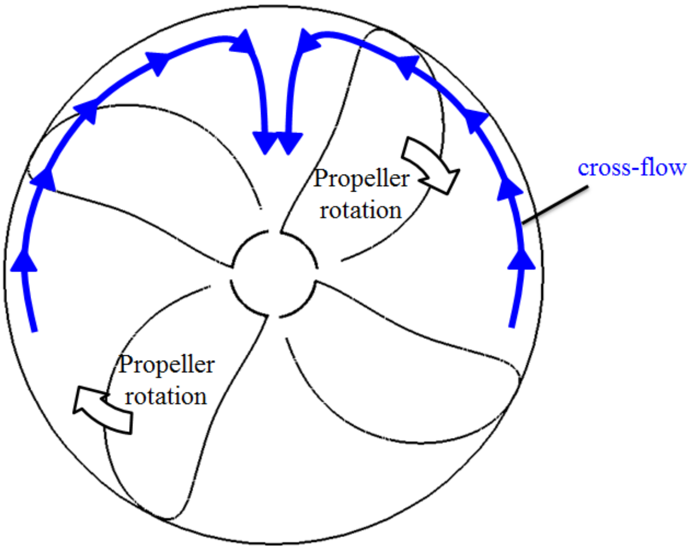

Figure 10: Schematic view of propeller rotation, and cross-flow.

The CFD results clearly displayed all the above flow features with and without PSS, confirming that the present CFD computations can be reliably utilized for the propulsive performance prediction with and without PSS.

5.2 Propulsion performance prediction for full scale

The wake fractions (w) and quasi-propulsive efficiency (η_D) of the full scale ship, which were predicted by the full scale CFD computation, ITTC 1978 and ITTC 1999, and the proposed method, are listed in Table 3. For comparison purposes, the full scale CFD results were considered as the reference values. The predicted values by ITTC 1978 method showed the largest difference. It is naturally understandable because the method does not take into account of ESD's effects at all. ITTC 1999 method, which is designed for ships with ESD's, showed better prediction. However, the final outcome, that is η_D, is still largely under-predicted. Also, due to the requirement for full scale computation, ITTC 1999 method is a high-cost way to predict the propulsive performance in full scale. The newly proposed method showed slightly over-predicted η_D. it may be due to ITTC 1957 method which is calibrated for general cargo ships without ESD's adopted for viscous component prediction. Or it may be able to the under-predicted η_D by the full scale computation. In other way, the difference is not significant, only 2.7%. Also it should be noted that the proposed method takes less computational resources and time only for model scale computations of the RANS and Euler equations.

Table 3: Wake fraction and quasi-propulsive efficiency by full scale CFD computations, and existing and proposed methods.

w/ pss	Method	w					η_D
		$w_{t,w/\ ESD,model}$	$w_{inv,w/\ ESD,model}$	$w_{v,w/\ ESD,model}$	$w_{v,w/\ ESD,\ full}$	$w_{t,w\ ESD,full}$	
Full scale computations	CFD					0.4865	0.753
Extension from model scale computations	ITTC 1978	0.5444	0.2043	0.3501	0.1585	0.3628	0.570
	ITTC 1999	0.5444	-	-	-	0.4360	0.644
	Proposed method	0.5444	0.3589	0.1955	0.1719	0.5308	0.774

6 SUMMARY AND CONCLUSIONS

To attain required EEDI, enhancement of the propulsion performance is one of the easiest and most efficient methods. Thus, interest on ESD's has increased recently. For ships with ESD's, extension of propulsive performance results from model to full scale ships is necessary.

The present study summarized the existing propulsive performance prediction methods and proposed a simple, reliable and efficient method with ESD's efficacy in mind. To prove utility of the proposed method, KVLCC2 with PSS was selected as the model problem. The existing and proposed methods were based on the CFD computations. Thus, CFD computations for the model and full scale ships were carried out. From results, PSS changed the direction of the cross flow, thus, was favorable to the thrust. The propulsive performance prediction in full scale was conducted by the existing and proposed methods and compared with that by the full scale CFD computations. The propulsive performance predicted by the proposed method was similar with that by the full scale computations. It was concluded that the proposed method could extend the model scale results to full scale ones with ESD's performance improvement effects.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (2013R1A1A2012597) funded by the Ministry of Education of the Korea government, the Multi-Phenomena CFD Research Center (20090093129) funded by the Ministry of Science, ICT and Future Planning of the Korea government, and Leaders Industry-university Cooperation (LINC) Project funded by the Ministry of Education, Science & Technology (MEST) and the National Research Foundation of Korea(NRF) of the Korea government.

REFERENCES

[1] Korkut, E., A case study for the effect of a flow improvement device (a partial wake equalizing duct) on ship powering characteristics. *Ocean Engineering* (2006) **33:** 205-

208.
[2] Celik, F., A numerical study for effectiveness of a wake equalizing duct. (2007) **34**: 2138-2145.
[3] Celik, F., Guner, M, 2007. Energy saving device of stator for marine propellers. *Ocean Engineering* (2007) **34**: 850-855.
[4] Hansen, H.R. Dinham-Peren, T. Nojiri, T., Model and full scale evaluation of a propeller boss cap fin device fitted to an aframax tanker, *2nd International Symposium on Marine Propulsors* (2011) Hamburg, Germany, 15-17 June
[5] Kawamura, T. Ouchi, K. Nojiri, T., Model and full scale CFD analysis of propeller boss cap fins (PBCF). *Journal of Marine Science and Technology* (2012) **17:** 469-480
[6] Lee, S.-K. Yu, K. Tseng, R.K.-C., Propeller performance of a containership fitted with energy-saving rudder fin in a seaway. *12th International Symposium on Practical Design of Ships and Other Floating Structures* (2007) Changwon, Korea.
[7] Choi, J.-E. Kim, J.-H. Choi, B.J. Lee, S.B. Chung, S.H. Seo, H.W., 2013. Development of energy-saving devices for a hull slow-speed ship through improving propulsion performance. *12th International Symposium on Practical Design of Ships and Other Floating Structures* (2007) Changwon, Korea.
[8] Shin, H.-J. Lee, J.-S., Lee, K.H. Han, M.R. Hur, E.B. Shin, S.C., Numerical and experimental investigation of conventional and un-conventional preswirl duct for VLCC. International Journal of Naval Architecture and Ocean Engineering (2013) **5**: 414-430.
[9] Hollenbach, U. Reinholz, O., Hydrodynamic trends in optimizing propulsion, *2nd International Symposium on Marine Propulsors* (2011), Hamburg Germany., 15-17 June.
[10] Park, J.S. Choi, Y.B. Kim, S.P., Kim, I.P. Hwang, Y.S. Park, J.J. Kim, Y.S. 2006. Pre-swirl stator structure for low speed vessel. Korean patent registered number: 10-2006-0108472.
[11] Shih, T.H. Liou, W.W. Shabbir, A. Yang, Z. Zhu, Z., A new k-ε eddy-viscosity model for high Reynolds number turbulent flows - model development and validation. *Computers & Fluids* (1995) **24(3)**: 227-238.
[12] Park, S. Park, S.W. Rhee, S.H., Lee, S.B. Choi, J.-E. Kang, S.H., Investigation on the wall function implementation for the prediction of ship resistance. International Journal of Naval Architecture and Ocean Engineering (2013) **5**: 33-46.
[13] Seo, J.H., Seol, D.M., Lim, T.G., Park, S.T., Han, B.W., Rhee, S.H., A stereo PIV measurement of a model ship wake in a towing tank with uncertainty assessment, *10^{th} International Symposium on Particle Image Velocimetry (PIV13)* (2013), Delft, The Netherlands.