Numerical simulation of the ship self-propulsion prediction using body force method and fully discretized propeller model

A S Bekhit
“Dunarea de Jos” University of Galati, Faculty of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania
E-mail: adham.bekhit@ugal.ro

Abstract. The study proposes a numerical solution for the self-propulsion prediction of the Japan Bulk Carrier (JBC) ship model based on two different approaches: the first is the body force method using an infinite-blade actuator disk, while the second is based on a 3-D fully discretized propeller using the sliding grid technique. The computations are performed for the ship with and without Energy Saving Device (ESD) by making use of the ISIS-CFD solver which is available in the commercial software Fine™/Marine provided by NUMECA. The solver is based on finite volume method to build the spatial discretization to solve the Reynolds-Averaged Navier-Stokes equation (RANSE) in a global approach. Closure to turbulence is made by making use of $k-\omega$ SST model for the actuator disk case and EASM for the propeller model case. A comprehensive investigation is brought into attention for the nominal wake and the effective wake flow, for clarifying and understanding the hull-propeller interaction. The computed results are compared to the experimental results presented in the Tokyo Workshop on the Naval Hydrodynamics 2015 and showed a reasonable agreement.

1. Introduction
Prediction of ship self-propulsion point is crucially important in the ship powering estimation process; it is stating the condition where the propeller thrust can equalize the total drag of the ship. Obviously, an accurate prediction of this condition will result in accurate power estimation. Previously, this process was achieved by either experimental or empirical based approach. However, these methods are either too expensive or lacking the capability to define the hull-propeller interaction from the hydrodynamic point of view. The hydrodynamic aspects of a marine propeller working behind the ship represent a very complicated challenge. Many problems are involved, including: flow separation, ventilation and cavitation. These problems can significantly affect the propeller performance and cause undesired consequences such as: loss of powering, noise, vibration and erosion. CFD has proven to be an efficient tool to predict the hull-propeller interaction associated problems, for model and full scale ships, besides, the complicated and special types propellers, for example [1-5]. The RANSE solvers development gave an alternative method to predict accurately the self-propulsion point and the hull-propeller interaction, as it was shown in the Workshops on CFD in Ship Hydrodynamics that were held in Gothenburg 2010 and Tokyo 2015 [6, 7]. In addition, the variety of the commercial software associated with the development in computation capacities resulted in high performance computations (HPC) made the CFD an alternative and cheap tool to solve ship propulsion problems.

Recently, the dominant methods implemented in CFD to solve the self-propulsion are the body force method and the fully discretized propeller. The former does not require modelling the propeller;
however, the body forces are applied on propeller location grid points, then the body forces are defined so that they integrate numerically to the thrust and torque of the propeller [8]. Another more complicated approach based on body force method, which is implemented in this paper, is to use a propeller performance code in an interactive fashion with the RANS solver to capture propeller-hull interaction and to distribute the body force according to the actual blade loading [9]. The latter is based on fully discretized propeller approach which involves the use of either sliding grid [10] or overset [11] techniques. It is worth mentioning that the body force method is simpler, cheaper and more common in the most recent researches, however, the fully discretized propeller can give a better definition for the blade loading and cavitation occurrence [12].

Based on the previous outline, this paper studies the self-propulsion prediction of the JBC ship model by the means of two different approaches: actuator disk method and fully discretized propeller. The analysis is performed for two cases when the ship is with and without the ESD. The JBC is a capesize bulk carrier equipped with a duct works as an ESD, was introduced as a benchmark for CFD investigations. The National Maritime Research Institute (NMRI), Yokohama National University and Ship Building Research Centre of Japan (SRC) were jointly involved in the design of the ship hull, the duct and the rudder. Towing tank experiments were planned at NMRI, SRC and Osaka University, which included resistance tests, self-propulsion tests and PIV measurements of stern flow fields [7]. Ship and propeller particulars are tabulated in table 1, while the geometry of the ship including duct and propeller is depicted in figure 1.

Table 1. JBC ship and propeller particulars.

| Ship Parameter               | Value   | Propeller Parameter               | Value   |
|------------------------------|---------|-----------------------------------|---------|
| Length between Perpendiculars, \( L_{pp} \) [m] | 7.0     | Diameter, \( D_p \) [m]           | 0.203   |
| Beam, \( B \) [m]            | 1.125   | Boss ratio, \( D_b/D_p \)         | 0.18    |
| Depth, \( D \) [m]           | 0.625   | Pitch ratio, \( P/D_p \)          | 0.75    |
| Draft, \( T \) [m]           | 0.4125  | Expanded area ratio, \( A_E/A_0 \) | 0.5     |
| Block Coefficient, \( C_B \) | 0.858   | Number of blades, \( Z \)         | 5       |
| Duct section                 | NACA4420| Blade section                     | AU      |
| Duct diameter, outlet        | 0.11165 | Direction of rotation             | clockwise|
| Duct opening angle           | 20\(^\circ\) | Angle of rake                     | 5\(^\circ\) |
| Propeller Location from A.P., \( x/L_{pp} \) | 0.014497 | Maximum blade width ratio         | 0.2262  |
| Propeller Location from base line, \( z/L_{pp} \) | 0.018507 | Blade thickness ratio             | 0.05    |

2. Mathematical model
The computations are performed using ISIS-CFD solver available by Fine\textsuperscript{TM}/Marine commercial software provided by NUMECA. The solver is based on the finite volume method to build a spatial discretization for the governing equation to solve the RANS equation in a global approach. Pressure-velocity coupling is enforced through a Rhie & Chow SIMPLE like method: at each time step, the velocity updates come from the momentum equation and the pressure is extracted from the mass conservation constraints transformed into pressure equation. Closure to the turbulence is achieved by making use of the \( k-\omega \) \textit{SST} model for the actuator disk approach and the EASM model for the discretized propeller approach. Free-surface is captured in an air-water interface based on the volume of fluid technique. All the computations are performed in an earth-fixed reference frame with only two degrees of freedom including heave and pitch.
2.1. Computational grids

All the computational grids are generated by the unstructured hexahedral grid generator Hexpress™. The computational domain for all the computations is of a rectangular configuration with a length five times the ship length, beam is four times the ship length and height is two times the ship length as depicted in figure 2.

![Image](image_url)

**Figure 1.** JBC ship model with duct and propeller.

**Figure 2.** Computational domain dimension and boundary conditions.

Boundary conditions are applied on the domain boundaries as well as on the solid wall as follows:
- inflow, located at $1.0L_{pp}$, Neumann boundary condition for the pressure; while, velocity and turbulent viscosity are imposed,
- outflow, located at $3.0L_{pp}$, the velocity components and turbulent viscosity are zero-extrapolated, while the pressure is having the static value,
- sides, located at $2.0L_{pp}$, all hydrodynamic parameters are extrapolated with zero-gradient,
- top and bottom, located at $0.5L_{pp}$ and $1.5L_{pp}$, respectively, the pressure is prescribed,
- ship solid wall, wall function having $y^+=30$, while the deck is having a slip condition; assuming that during the computation it remains in the air while the viscous effect can be neglected.

Besides, for the fully discretized propeller, a slip condition was applied on the shaft, hub and boss cap. The free-surface is initiated at the design draft located at $0.4125$m from the base line as highlighted in blue in figure 2.
The computational grids were generated for the actuator disk approach for case without and with ESD having 5.148 M and 5.322 M cells, respectively. Obviously, the difference in the number of cells is due to the addition of the duct. Similarly, for the fully discretized propeller, the number of cells for case without ESD was 11.048 M cells distributed as 5.359 M on the propeller and 5.689 M on the hull. On the other hand, for case with ESD, the number of cells was 12.482 M cells distributed as 6.359 on the propeller and 6.123 M cells on the ship. The difference in this case regarding the total number of cells is due to the change in the sliding grid configuration to avoid interference with the ESD. The four meshes are depicted in figure 3.

![Figure 3](image)

**Figure 3.** Discretisation grids for: (a) actuator disk model, (b) propeller model, without ESD (top) and with ESD (bottom).

To insure the grid similarity for both case studies with and without ESD, similar refinement variables for the domain, ship and wake zone were implemented. An isotropic refinement to capture the wake flow is applied for the wake zone for both approaches, while for the actuator disk approach; an extra refinement on the actuator disk was added to insure 25 grid cells at the disk thickness and 35 grid cells at the radial section as depicted in figure 3.

In the actuator disk simulation, the computation is performed based on a steady quasi-static approach to accelerate the flow within a time-span corresponding to the relationship between the length of the ship and its inflow velocity such that $T_{acc} = 2.0L_{pp}/U_{\infty}$. Computations are performed for 30 seconds using 10 nonlinear iterations per time step. The integration in time is done based on a second order convergence criteria using combined upwind discretization scheme and centered discretization scheme. The computation technique is based on using the open water data from the tank test or even
from a CFD computation to update the actuator disk rotation after the nominal wake is estimated in every time step. After the forces are balanced, the thrust identity is used to predict the self-propulsion point for the propeller working in the ship wake.

On the other hand, for the fully discretized propeller, unsteady simulation is performed which was divided in two consecutive computations. The first computation is meant to stabilize the ship in its hydrodynamic position using a ship based time step as in case of the simple resistance computations, in this case a large time step is used, only 5 nonlinear iteration and a second order convergence criteria are imposed. The second one is to stabilize the propeller thrust using a time step based on the propeller rotation to provide 200 time steps per propeller rotation. Nonlinear iterations are increased for this computation to 12 with third order convergence criteria to improve the accuracy of results. The simulation is started with the estimated propeller rotation obtained from the tank test. The analysis of forces is then applied to satisfy the condition where; \( T + SFC - R = 0 \), while \( T \) is the propeller thrust, \( R \) is the total resistance and the SFC is the towing force used during the tank test. If this condition is not achieved, one or more extra computations can be performed imposing a different propeller rotation, and finally the condition can be satisfied by interpolating the results obtained from these different computations. It is worth mentioning that this technique is the one implemented for the presented computations using three different imposed propeller rotations, as it will be discussed in the results section.

Simulations are performed for the actuator disk on a 16 cores Xeon CPU at 3.47 GHz and 96 GB of RAM. Convergence was achieved within 56.6 hours of computation, while the fully discretized simulation was performed on an HPC concept with 120 processors at 3.3 GHz and 128 GB of RAM. Convergence was achieved for the four computations: the ship acceleration computation plus the three computations required for extrapolation within approximately 193.3 hours. To get a flavour of the comparison between the two approaches, the actuator disk computation was repeated on the HPC with the same number of processors and the convergence was achieved within 18.9 hours.

3. Results and discussion

3.1. Actuator disk method

For the actuator disk, it is essential to compute the total resistance and the nominal wake as a primary step for the computation. For results consistency, the nominal wake was computed for the ship with and without the ESD on the same grid for the actuator disk approach and results are compared with the experimental results summarized in [7,13]. The comparison between the computed wake and the measured one is brought into attention in figure 4.a and b for ease without and with ESD, respectively. The wake is measured at the propeller centre plane distanced 0.10148m from the after perpendicular. The results showed that the computed wake is in a reasonable agreement with the measured one, though it seems to be slightly under predicted; a fact that might tend to affect the effective wake computations, as it will be discussed later.

The results for the self-propulsion coefficients computed based on the actuator disk method are tabulated in table 2 for case with and without ESD. The comparison between the computed coefficients and the measured ones in the tank test shows a reasonable agreement. The error between the computed and the measured results is computed based on: \( E\%D = 100 \times (EFD-CFD)/EFD \). The effect of the ESD seems to be promising considering the fact that the thrust and torque coefficients are enhanced and the thrust deduction fraction and wake fractions are reduced. Besides, the hull efficiency is also improved. Obviously, the aforementioned under prediction of the nominal wake had that influence on the wake fraction coefficient \((1-w)\) as it can be noticed in the results. This may explains the higher error value associated with the wake fraction coefficient.
Figure 4. Computed axial flow velocity contours at the propeller plane for ship: (a) without ESD, (b) with ESD compared to the EFD [13].

Table 2. Self-propulsion coefficient computed for ship with and without ESD based on actuator disk method.

| Coefficient                              | Without ESD | E%D | With ESD | E%D |
|------------------------------------------|-------------|-----|----------|-----|
| Total resistance coefficient, $C_T \times 10^3$ | 4.81        | 4.93% | 4.76     | 4.28% |
| Thrust coefficient, $K_T$                | 0.217       | 0.64% | 0.233    | 0.086% |
| Torque coefficient, $10K_Q$              | 0.279       | 1.57% | 0.295    | 0.915% |
| Advance coefficient, $J$                 | 0.410       | 0.68% | 0.36     | 2.75% |
| Thrust deduction coefficient, $(1-t)$    | 0.803       | 1.11% | 0.810    | 1.97% |
| Wake fraction coefficient, $(1-w)$       | 0.552       | 6.30% | 0.471    | 3.06% |
| Relative rotative efficiency, $\eta_r$   | 1.011       | 1.08% | 1.014    | 1.38% |
| Hull efficiency, $\eta_H$                | N.A         | -    | N.A      | 1.739 |

The effective wake downstream the propeller at the after perpendicular is depicted in figures 5.a and b for ship without and with ESD, respectively. The effective wake can be divided in two different zones; the flow within the propeller region and the flow outside the propeller region. The flow inside the propeller is showing an asymmetric configuration due to the propeller rotation resulting in a moon crescent-like region of a high velocity [14].

Figure 5. Computed axial flow velocity contours at the after perpendicular for ship: (a) without ESD, (b) with ESD compared to the EFD [13].

The axial flow velocity contours seem to be under predicted; this is due to the aforementioned under prediction estimated in the nominal wake. The overall under prediction of the wake might be highly affected by the turbulence model; considering the fact that the JBC is a full ship with a very
high block coefficient; the wake for these types of ships is quite large. A further investigation with a different turbulence model might be necessary. Another reason for the under prediction of the wake might be related to the mesh density in the wake area. This may also require a different investigation imposing an extra refinement in the wake zone.

Despite the fact that the actuator disk method is quite effective, easier and faster, the lack of understanding the flow around the propeller and pressure distribution on the propeller blades is not applicable in this method. This is why the need for the fully discretized propeller method is necessary to study the hull-propeller interaction, especially when certain phenomena are to be investigated such as propeller cavitation or individual blade loading. In the next section, more details about the propeller model are described, not taking into account the cavitation phenomenon.

3.2. Fully discretized propeller
The computation requires three separate attempts to help predicting the propeller revolution rate that tends to justify the force equilibrium condition as mentioned in the previous section. The results interpolation for case with and without the ESD is plotted in figure 6. The measured revolution rate for the self-propulsion point was given for the ship without ESD as 7.8 revolutions per second (rps) while for ship with ESD was 7.5 rps. The predicted revolution rate is slightly under predicted as it can be noticed from the diagram. Error for case without ESD compared to the measured value is about 0.99%, while for case with ESD is about 1.531%.

![Figure 6. Results interpolation to predict the propeller revolution rate.](image)

Similarly, the computed self-propulsion coefficients for the fully discretized propeller model are tabulated in table 3 and compared with the EFD results. The results seem to be slightly over predicted for both cases with and without the ESD. Results for case with ESD show a reasonable agreement in comparison with the measured values; however, for case without ESD it over predicts the forces significantly for resistance and thrust. This may require a proper investigation on a finer grid to enhance the computed results.

The pressure distribution on the propeller blades is depicted in figure 7. A stagnation effect can be noticed on the leading edge of the propeller blades, as expected, which results in a high pressure. The aforementioned asymmetric velocity distribution is expected to occur as an effect on the pressure distribution on the both sides of the propeller blades. The pressure is reaching a minimum value heading towards the propeller tip, while another pressure drop occurs on the hub tip; those both pressure drops are causing a velocity increment that causes the tip and hub vortices in the propeller wake, as it will be discussed later.
Table 3. Self-propulsion coefficient computed for ship with and without ESD based on 3-D propeller model method.

| Coefficient                          | Without ESD | With ESD |
|--------------------------------------|-------------|----------|
|                                      | EFD         | CFD      | E%D     | EFD   | CFD     | E%D     |
| Total resistance coefficient, $C_T \times 10^3$ | 4.811       | 4.913    | 2.12    | 4.76   | 4.685   | 1.576   |
| Thrust coefficient, $K_T$            | 0.217       | 0.232    | 6.912   | 0.233  | 0.2427  | 4.16    |
| Torque coefficient, $10K_Q$          | 0.279       | 0.291    | 4.30    | 0.295  | 0.306   | 3.72    |
| Advance coefficient, $J$             | 0.410       | 0.414    | 0.98    | 0.36   | 0.366   | 1.67    |

Figure 7. Computed pressure distribution on the propeller blade suction side (left) and pressure side (right), coloured by non-dimensional pressure coefficient.

3.3. Interaction between hull and propeller

The wake region where the propeller works is a region with significant hydrodynamic phenomena. The propeller working in this region is definitely affected by the flow separation caused by the hull form, which tends to reduce the propulsion efficiency. Yet, the propeller itself is causing major changes in the wake zone due to the propeller rotation causing the helical structure of the flow known as vortices. The vortical structure of a propeller wake, as was mentioned before, is a result of an imbalance between pressure and velocity that occurs usually nearby the propeller tip causing the tip vortices, at the propeller hub resulting in hub vortices and finally at the propeller root causing root vortices. The vortical structure of the wake region is given in figure 8; which depicts the vortex structure of the wake flow measure by the second invariant for the iso-surface $Q=30$ and coloured based on the non-dimensional helicity. Figure 8.a depicts the hull induced vortices which is generated due to the flow separation at the bilge rise starting from the bilge and heading downstream. Figure 8.b shows the vortices computed based on the actuator disk approach. The vortex starts at the tip of the disk and is washed downstream due to the viscous diffusion, while the bilge vortices generated on the both sides of the hull are combined together due to the disk rotation. The modelled propeller vortical structure is depicted in figure 8.c; the vortices are influenced, as expected, by the bilge vortices, which tend to continue past the propeller due to the suction effect of the propeller. The tip and hub vortices can be noticed clearly in the propeller refinement region where the sliding grid takes place, then vanish quickly afterwards or even combined with other vortices and continue downstream. The vortical structure of the propeller model shows that the mesh is not fine enough to capture the vortices downstream, as the root vortices seem to be missing. This is also requires enhancing the mesh refinement in the propeller region.
Figure 8. Computed second invariant at Q=30 iso-surface coloured by non-dimensional helicity for: (a) bare hull, (b) actuator disk and (c) propeller model; without duct (top), with duct (bottom).

An extra investigation for the flow before and after the propeller is brought into attention in figure 9. The axial velocity contours are depicted for two different locations especially chosen to investigate the effect of the duct; the first is locate between duct and propeller at a distance 0.11m from the after perpendicular, while the second location is chosen directly after the propeller at a distance 0.05m from the after perpendicular. It can be noticed that the flow speed before the propeller is increased in case with duct, which tend to enhance the propeller performance. The flow velocity downstream the propeller is also increased due to the linear and angular momentum gained from the propeller rotation.

Figure 9. Computed axial flow velocity contours before (top) and after (bottom) the propeller plane for ship: (a) without ESD, (b) with ESD.

As a consequence for the propeller existence in the wake, the free-surface can be affected by the propeller rotation causing a rise in the free-surface elevation and asymmetric configuration on the free-surface view. This influence might be significant when the propeller tip is close to the free-surface [14]. The effect of the propeller in this computation can be visualized in figure 10, where the
comparison between the bare hull free-surface topology and the hull with the propeller computed based on the actuator disk approach on the same grid is compared showing the slight change in the wave height at the stern region besides an asymmetric distribution of the wave contours at the stern region. Note that the wave elevation is corresponding to the initial undisturbed free-surface located at 0.4125m from the baseline.

![Figure 10](image_url) Comparison between the computed free-surface topology with propeller (left) and without the propeller (right).

4. Concluding remarks
The self-propulsion prediction of the Japan bulk carrier ship model with and without the energy saving device is presented and solved based on two different approaches; the first is the body force method using an infinite blade actuator disk, while the second is based on using a fully discretized propeller model. Both models showed their capability of predicting the self-propulsion point of the ship. Nevertheless, every method has its own pros and cons. On one side, the actuator disk showed to be simple, easier and faster to compute the self-propulsion coefficients; nevertheless, the lack of understanding the real flow downstream the propeller, propeller blade pressure distribution and the cavitation occurrence is not applicable in such a technique. Yet, it can still stand as a sufficient tool for a quick prediction of the self-propulsion coefficients for the ship in early design stage or to study the implementation of wake equalizing tools such as ducts or stern fins to optimize the propeller performance. On the other side, the propeller model showed to be efficient, reliable and all the flow characteristics and blades pressure can be fully visualized; however, it takes a significant computational time and is hard to model and compute.

The self-propulsion parameters computed based on the two approaches showed a reasonable agreement with the experimental results. The error between computed and measured results for the actuator disk approach was within 0.64 and 1.57% for thrust and torque coefficients, respectively; yet, a lack of accuracy accompanied with the wake flow prediction was encountered for both cases with and without the ESD with an error 6.3 and 3.06% for cases without and with ESD, respectively. This lack can be related to the efficiency of the turbulence model used for computation or even form the accumulated error based on the nominal wake prediction.

For the propeller model, predicting the self-propulsion point was satisfactory. The error for the self-propulsion point revolution rate was 0.99 and 1.53% for cases without and with ESD, respectively; however, a relatively significant error was recorded for predicting the thrust and torque coefficient that ranged between 3.72 and 6.912%. This might require a further investigation on a finer grid to predict efficiently the reason behind this deviation.

Analysis of hull propeller interaction was presented and showed that both methods can predict the hull propeller interaction correctly. The propeller performance is affected by the flow separation; which was enhanced by imposing the ESD. Besides, the propeller induced axial and angular momentums are affecting significantly the flow characteristics past the propeller.
Using the ESD before the propeller helped enhancing the flow characteristics upstream the propeller by enhancing the velocity of the inflow and also helped to reduce the large wake generated by the ship which has significant form coefficients.

Though the validation of results seems to be qualitatively and quantitatively promising, a further verification study is crucially important to gain more details about the computational model consistency and to understand clearly the hull-propeller interaction. This step is set to be performed in the very near future by the author.

5. References
[1] Castro A M, Carrica P M and Stern F 2011 Full scale self-propulsion computations using discretized propeller for the KRISO container ship KCS J. Computers & Fluids 51 35–47
[2] Krasilnikov V I 2013 Self-propulsion RANS Computations with a single-screw container ship Proc. 3rd Int. Symp. on Marine Propulsors SMP13 (Launceston, Tasmania, Australia, 5-8 May 2013) pp 430–438
[3] Ponkratov D, Čegos C 2015 Validation of ship scale CFD self-propulsion simulation by the direct comparison with sea trials results Proc. 4th Int. Symp. on Marine Propulsors SMP15(Austin, Texas, USA, 31 may – 4 June 2015) pp 289–299
[4] Pacuraru F, Lungu A, Ungureanu C and Marcu O 2010 Numerical simulation of the flow around a steerable propulsion unit IOP Conf. Ser.: Earth Environ. Sci. 12(1) no. 012032 (IOP Publishing Ltd)
[5] Pacuraru F, Lungu A, Marcu O 2011 Self-propulsion simulation of a tanker hull AIP Conf. Proc. 1289(1) 191-194
[6] Larsson L, Stern F and Visonneau M 2013 Numerical Ship Hydrodynamics, an Assessment of the Gothenburg 2010 Workshop (Netherlands: Springer)
[7] National Maritime Research Institute (NMRI) 2015 A Workshop on CFD in Ship Hydrodynamics (Tokyo, Japan) http://www.t2015.nmri.go.jp
[8] Stern F, Kim H T, Patel V C and Chen H C 1988 A Viscous flow approach to the computation of propeller-hull interaction J. Ship Research 32(4) 246–262
[9] Stern F, Kim H T, Zhang D H, Toda Y, Kerwin J and Jessup S 1994 Computation of viscous flow around propeller-body configurations: series 60 CB = 0.6 ship model J. Ship Research 38(2) 137-157
[10] Queutey P, Deng G B, Wackers J, Guilmineau E, Leroyer A and Visonneau M 2012 Sliding grids and adaptive grid refinement for RANS simulation of ship-propeller interaction J. Ship Technology Research 59(2) 44-58
[11] Carrica P M, Castro A M and Stern F 2010 Self-propulsion computations using a speed controller and a discretized propeller with dynamic overset grids J. Marine Science Technology 15 316-330
[12] Carrica P M, Castro A M and Stern F 2010 Self-propulsion computations using a speed controller and a discretized propeller with dynamic overset grids J. Marine Science Technology 15 316-330
[13] Stern F, Yang J, Wang Z, Sadat-Hosseini H, Mousaviraad M, Bhushan S and Xing T 2012 Computational Ship Hydrodynamics: Nowadays and Way Forward J. Int. Shipbuilding Progress 60(1-4) 3-105
[14] Hirata N 2015 http://www.t2015.nmri.go.jp/Presentations/Day1-AM2-JBC-TestData1-Hirata.pdf
[15] Lungu A 2018 Numerical simulation of the resistance and self-propulsion model tests Proc. the ASME 2018 37th Int. Conf. on Ocean, Offshore and Arctic Engineering OMAE2018 OMAE2018-77767 (Madrid, Spain, 17-22 June 2018) (under press)