Gemi ve Deniz Teknolojisi Dergisi
Sayı: 219, Haziran 2021
ISSN: 1300-1973, e-ISSN: 2651-530X, Dergi ana sayfası: http://www.gmoshipmar.org/
Araştırma Makalesi

Deplasman Tipi Gemiler için Sayısal Analizlerin Gerçeklemesi ve Doğrulaması

Sarih Sarı[1], Ali Doğrul[2]

[1] Gemi İnşaatı ve Gemi Makineleri Mühendisliği Bölümü, Yıldız Teknik Üniversitesi, İstanbul, Türkiye
[2] Gemi İnşaatı ve Gemi Makineleri Mühendisliği Bölümü, Deniz Harp Okulu, Milli Savunma Üniversitesi, Tuzla, İstanbul, Türkiye

[1] (sorumlu yazar), sarih@yildiz.edu.tr, ORCID: 0000-0002-0665-5046
[2] adogrul@dho.edu.tr, 0000-0003-3236-555X

ÖZET

Gemilerde güç ihtiyacı ve yakıt tüketimini en aza indirmek çok önemlidir, bu sayede daha çevreci gemiler elde edilmiş olur. Bu hedef, gemilerin hidrodinamik performansının tahmin edilmesi ile elde edilebilir. Bu bağlamda, çeşitli araştırmacılar tarafından deneysel ve sayısal yöntemler yaygın olarak kullanılmaktadır. Deneysel çalışmalar model deneylerine dayanırken sayısal yöntemler viskoz ve potansiyel akış kabullerine dayanmaktadır. Bu çalışmada çeşitli tipte gemiler sayısal olarak incelenerek hem gerçekleme hem de doğrulama adına kapsamlı bir veri setinin sunulması amaçlanmıştır. Sayısal yöntemin doğruluğunu ve hassasiyetini göstermek adına iki konteyner gemisi ve bir muharip suüstü gemisi için RANS denklemlerini çözen sayısal bir yaklaşım kullanılmıştır. Bu gemiler KRISO konteyner gemisi (KCS), Duisburg test gemisi (DTC) ve ONR (Office of Naval Research) tarafından geliştirilen teknedir. Bu gemiler etrafındaki akış incelenirken serbest yüzey etkileri hesaba katılmıştır. Akış analizleri sakin su koşullarında gerçekleştirilmiştir ve gemiler paralel batma ve trim hareketine karşı serbest bırakılmıştır. Belirsizlik çalışması için ITTC ve AIAA tarafından önerilen GCI yöntemi kullanılmıştır. Sık, orta ve seyrek olacak şekilde ağ boyutu ve zaman adımı açısından farklı analiz setleri kurgulanmıştır. Bu analiz setleri sabit bir iyileştirme oranıyla $(\sqrt{2})$ oluşturulmuştur. Belirsizlik amaçlı sayısal analizler her bir geminin dizayn Froude sayısında gerçekleştirilmiştir. Belirsizlik değerleri toplam direnç açısından elde edilmiştir. Bunu takiben, düşük ve orta hızları kapsayacak şekilde geniş bir Froude sayısı aralığında her bir model gemi için kapsamlı bir doğrulama çalışması yapılmıştır. Doğrulama, sayısal sonuçların erişilebilen deneysel sonuçlarla kıyaslanmasıyla yapılmıştır. Buna ek olarak, sonuçlar literatürde mevcut diğer sayısal sonuçlarla karşılaştırılmıştır. Doğrulama, toplam direnç, paralel batma ve trim açısı parametreleri üzerinden yapılmıştır. Bu çalışma, hesaplamalı akışkanlar dinamiği (HAD) yönteminin gemi hidrodinamik performansını yeterli düzeyde tahmin edebildiğini göstermiştir. Bu sonuçlara göre karşılaştırma için deneysel veri eksikliğinde sayısal yöntem düşük belirsizlik değerleriyle güvenilirdir.

Anahtar kelimeler: Karşılaştırma, HAD, RANS, Toplam Direnç, Belirsizlik, Doğrulama.

Makale geçmişi: Geliş 24/04/2021 – Kabul 28/06/2021

GMO Journal of Ship and Marine Technology Journal
Volume: 219, June 2021
ISSN: 1300-1973, e-ISSN: 2651-530X, Journal homepage: http://www.gmoshipmar.org/
Research Article

Verification and Validation of Numerical Simulations of Displacement Type Vessels

Sarih Sarı[1], Ali Doğrul[2]

[1]Department of Naval Architecture and Marine Engineering, Yildiz Technical University, Istanbul, Turkey
[2]Department of Naval Architecture and Marine Engineering, Turkish Naval Academy, National Defence University, Tuzla, Istanbul, Turkey

[1] (corresponding author)sarih@yildiz.edu.tr; ORCID: 0000-0002-0665-5046
[2]adogrul@dho.edu.tr; ORCID: 0000-0003-3236-555X

ABSTRACT

It is crucial to reduce the power need and fuel consumption of ships, thus eco-friendly ship design can be achieved. This goal can be achieved with the accurate prediction of the hydrodynamic performance of ships. In this manner, numerical and experimental methods are widely used by many researchers. Experimental studies are based on towing tank tests while the numerical methods are based on viscous and potential flow assumptions. In this study, it is aimed to investigate different types of ship models to provide a comprehensive data set. A numerical approach solving RANS (Reynolds-averaged Navier-Stokes) equations was employed for two container ships and a naval surface combatant to show the precision and accuracy of the numerical method. These vessels are KRISO container ship (KCS), Duisburg test case (DTC) and ONR Tumblehome (ONRT) developed by the Office of Naval Research. The flow around these vessels was investigated by taking the free surface into account. The flow analyses were carried out in calm water conditions and the ships were set to be free to sinkage and trim. For the verification study, the GCI method, which is recommended by ITTC (International Towing Tank Conference) and AIAA (American Institute of Aeronautics and Astronautics), was employed. Fine, medium and coarse cases were generated with different grid sizes and time step sizes. These cases were generated by using a constant refinement ratio $(\sqrt{2})$. The numerical analyses for the verification purpose were conducted at the design Froude number of each model ship. The uncertainty values were obtained for the total resistance. Following this, a comprehensive validation study was conducted for each ship model in a wide range of Froude numbers, covering low and moderate speeds. The validation was done by comparing the numerical results with the available experimental data. In addition to this, the results were compared with other existing numerical results in the literature. The validation was done in terms of total resistance, sinkage and trim parameters. This study showed that the computational fluid dynamics (CFD) method can sufficiently estimate the ship's hydrodynamic performance. Within these results, when there is a lack of experimental data for comparison, the numerical method is again reliable having low spatial and temporal uncertainty values.

Keywords: Benchmark, CFD, RANS, Total Resistance, Uncertainty, Validation*.*

Article history: Received 24/04/2021 – Accepted 28/06/2021

1. Introduction

The hydrodynamics of displacement type vessels is an essential field to study to obtain optimum hull forms. Within this aspect, the hydrodynamic performance of these vessels is studied by using several methods. The model experiments are conducted to measure the total resistance of the ship in the model scale. Computational methods are also widely used for the prediction of hydrodynamic performance of various ship types. These methods are the potential flow and viscous methods. The potential method is suitable to observe wave resistance and wave patterns while neglecting the viscous effects. On the other hand, RANS (Reynolds-Averaged Navier-Stokes) solvers based on the viscous flow are used by researchers. The flow field around the hull can be modeled considering viscous and free surface effects together.

Several researches were made to predict the total resistance and power of the ships. For this purpose, benchmark ship models were designed and tested in towing tanks around the world. These model experiments provide significant experimental data for validation purposes. The benchmark models are mostly displacement type ships and calm water resistance data of these models can be found easily in the open literature. Thus, several numerical and experimental studies focusing on validation and numerical prediction of ship hydrodynamics have been carried out. For this purpose, Korean Research Institute of Ships and Ocean Engineering (KRISO) have developed container and crude rarrier type ships. KCS AND KVLCC2 (KRISO Very Large Crude Carrier) hulls were tested comprehensively in a towing tank and the flow field was observed in terms of various parameters. Total resistance, wake profiles along the hull and the nominal wakefields were obtained for various Froude numbers (Kim et al., 2001).

KCS and KVLCC2 hulls were analyzed for calm water resistance prediction in different scales (Can et al., 2020; Dogrul et al., 2020; Ozdemir et al., 2016; Pereira et al., 2017; S. Van et al., 2011; S. H. Van et al., 2006; Zhang, 2010). Besides, the self-propulsion performance of these models was also calculated by several studies (Carrica et al., 2011; Shen et al., 2015).

Another container vessel was designed in University of Duisburg-Essen to investigate the flow field and obtain detailed data in various topics such as calm water resistance and self-propulsion, seakeeping. Duisburg Test Case (DTC) model was first investigated to estimate total resistance and self-propulsion performance experimentally (Moctar et al., 2012). This model was mostly used for the prediction of hydrodynamic performance in confined and/or shallow water numerically (Kok et al., 2020; Terziev et al., 2018; Tezdogan et al., 2016). Also, a recent experimental study was conducted for the prediction of self-propulsion characteristics in a different model scale (Kinaci et al., 2020).

ONR Tumblehome designed by the Office of Naval Research is a pre-design form of Zumwalt class US destroyer. A comprehensive experimental study of ONRT was performed in calm water for both bare and appended hull forms (Cook, 2011). Another experimental study was conducted for the bare hull for additional Froude numbers to obtain calm water hydrodynamic parameters (C. Delen & Bal, 2019). A numerical study was carried out in different scales including the full scale to determine the resistance and self-propulsion characteristics of ONRT (C. Delen et al., 2020). Another numerical study was conducted to investigate the hydrodynamics of ONRT by captive model analyses (Guo et al., 2018).

In this study, three benchmark ships were investigated in model scale. These benchmark models were chosen as KCS, DTC and ONRT hulls. The numerical analyses were performed using the RANS method. Firstly, the numerical approach was verified using the GCI method through grid and time step for all ship models. The numerical uncertainty values of all three ship models were presented in terms of grid spacing and time step size. Following this, the numerical analyses were extended to several Froude numbers. The results were validated against various numerical and experimental studies through total resistance coefficient, sinkage and trim values. The numerical results of KCS model were compared with the results of different experimental data. DTC model was compared in terms of total resistance because of the lack of experimental data of trim and sinkage. ONR Tumblehome model was analyzed and the results were compared with different experimental and numerical results. These numerical/ experimental results belong to different solvers/towing tanks having different bias errors and cover different ranges of Froude numbers. The present study covers a wider range of Froude numbers for the ONRT hull. As a concluding remark, a comprehensive numerical data was provided to the literature for three types of ship hulls including the numerical verification in terms of spatial and temporal uncertainty.

2. Theoretical Background

Since the flow phenomenon investigated in this study is time-dependent, unsteady RANS approach was employed for the analyses. For the uncertainty assessment, the numerical uncertainty values were calculated both spatially and temporally. The numerical approach and the verification procedure were explained in the following sub-chapters. The total resistance was decomposed into its components and the frictional resistance was also compared with the empirical one calculated by ITTC 1957 formulation.

2.1 URANS Approach

A commercial CFD software solving unsteady Reynolds-Averaged Navier-Stokes (URANS) equations was used in the numerical analyses. The continuity equation and the momentum equations are the governing equations. The flow is considered incompressible and turbulent. The continuity equation can be given as:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

The mean momentum equation is given in tensor notation with Cartesian coordinates:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\upsilon\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{2}$$

Here, ρ represents the density while U_i is the velocity vector and P stands for the pressure. The last two terms represent the viscous stress tensor and Reynolds stress tensor while υ is the kinematic viscosity.

A computational domain was created to simulate the flow around the ship model. The finite volume method (FVM) was used for the domain discretization and the governing equations were solved. A first-order temporal discretization and a second-order upwind convection scheme were applied in solving the momentum equations. SIMPLE type solution algorithm was employed.

The turbulence in the flow was modeled using the k-ε turbulence model which is very common in ship hydrodynamics applications (Tezdogan et al., 2015). The details about the turbulence model can be found in several references (Menter, 1994; Menter, 2009; Wilcox, 2008).

This study focuses on the flow around different ship models moving in calm water. For this reason, the free surface and the interaction between the air and water phases were modeled by utilizing the Volume of Fluid (VOF) method with the HRIC (High Resolutiın Interface Capturing) technique (Ferziger et al., 2020).

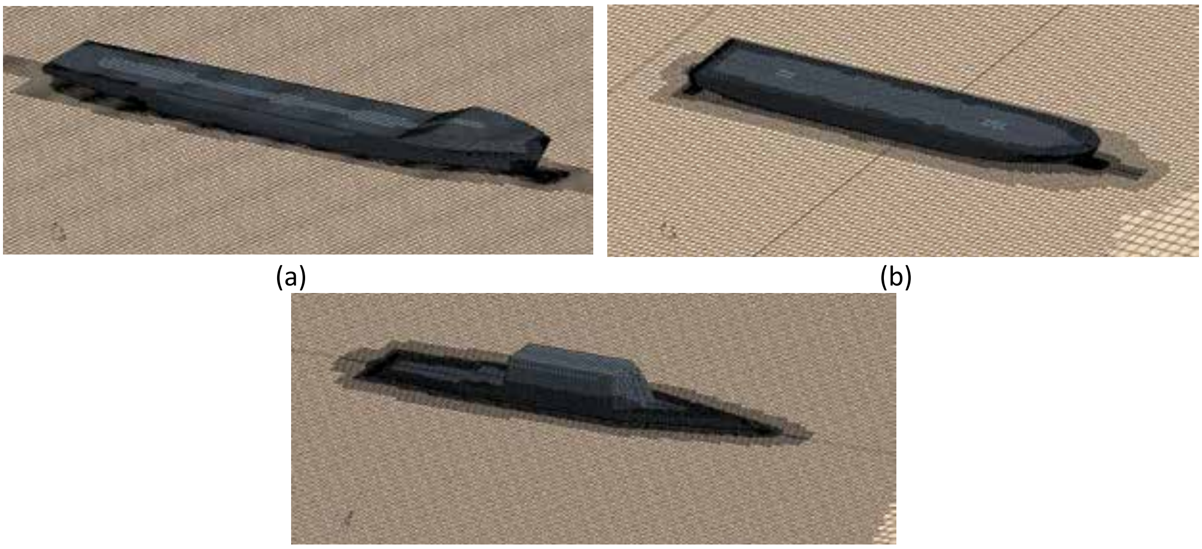
2.2 Numerical Method

The computational domain dimensions were chosen following the ITTC recommendations (ITTC, 2014a). Half of the ship was modeled to minimize the computing time because the ship models are designed axisymmetric. Since the free surface effects were taken into account, the inlet, top and bottom surfaces were defined as velocity inlet. The outlet was set to be pressure outlet. The side surfaces were defined as symmetry planes while the surface of the ship model was defined as no-slip wall. The computational domain was created in accordance with the ITTC recommendations (ITTC, 2014a). The computational domain dimensions are $2L_{PP}$ in the upstream and $3L_{PP}$ in the downstream direction. The width of the domain is $1.5L_{PP}$ while the domain is modeled with the half of the model. And the total height of the domain is $3L_{PP}$.

The computational domain was discretized with finite hexahedral elements using the trimmer mesh algorithm. Besides, local mesh refinements were made around the hull, wake region and free surface. The prism layer near the hull surface was modeled with caution to keep the wall y^+ values between 30 and 300. Figure 1 shows the grid structure applied on the computational domain and hull surfaces.

$$y^+ = \frac{yu_\tau}{\upsilon} \tag{3}$$

Here, y is the first cell size near the wall, u_τ is the frictional velocity and υ is the kinematic viscosity.



(a) (b) (c)

Figure 1. Grid structure applied for KCS (a), DTC (b) and ONRT (c).

2.3 Grid Convergence Index (GCI)

The uncertainty assessment was conducted using the GCI method through time step and grid sizes. GCI method which is based on Richardson extrapolation (Richardson, 1911) was first proposed by

Roache (Roache, 1998) and the revised procedure (Celik et al., 2008) is commonly used for uncertainty studies. Grid Convergence Index method is recommended by ITTC (ITTC, 2014b) and AIAA (Cosner et al., 2006) for the numerical uncertainty calculations. The details of the method can be found in the literature (Celik et al., 2008; Xing and Stern, 2010).

The numerical uncertainty was obtained by using three different cases of grid resolution and time step. Fine, medium and coarse analysis cases were generated using a refinement ratio of $\sqrt{2}$ that is also used by similar recent studies (Dogrul et al., 2020; Kahramanoglu et al., 2020; Sezen et al., 2018). The solution scalar in the calculation was chosen as the total resistance and the numerical uncertainty values were calculated at the design speed of each ship model. The uncertainty results can be found in the next chapter.

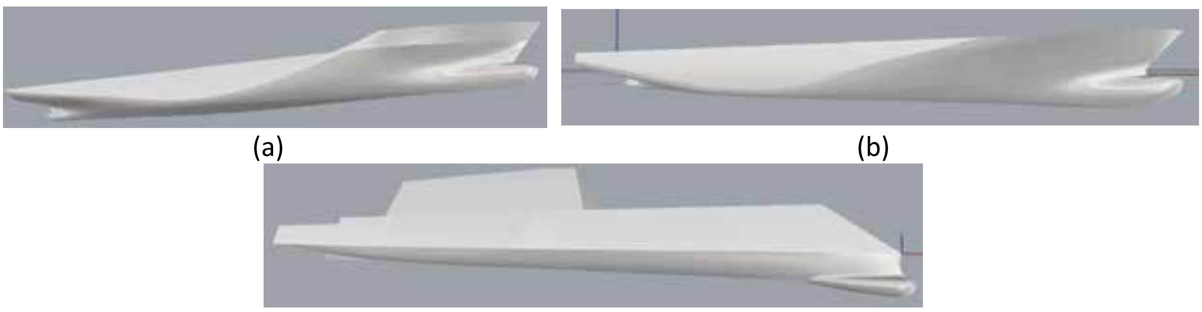
3. Ship Main Particulars

In this study, the well-known benchmark vessels were investigated numerically in model scale. These vessels are the KRISO container ship, Duisburg Test Case and ONR Tumblehome. The main particulars and 3-D views of each model were presented.

The main particulars of each ship in the model scale are given in Table 1. Figure 2 presents the 3-D views of the models. All three models have a bulbous bow and transom stern geometry. ONRT model has its superstructure however the aerodynamic effects were neglected in this study. The first two models are container ships while the last one is a surface combatant that is a pre-contract design of Zumwalt class destroyer (Cook, 2011).

Table 1. Main particulars of the benchmark ship models.

Main Particular	Symbol	KCS	DTC	ONRT
Scale	λ	31.60	59.407	48.935
Length	L (m)	7.3575	5.976	3.147
Beam waterline	B_{WL} (m)	1.0190	0.859	0.384
Draught	T (m)	0.3418	0.244	0.112
Displacement	Δ (m^3)	1.6489	0.827	0.0727
Wetted surface area	S (m^2)	9.5117	6.243	1.345
Block coefficient	C_B	0.651	0.661	0.535



(a) (b) (c)

Figure 2. Perspective view of KCS (a), DTC (b) and ONRT (c).

4. Numerical Results

Numerical analyses were performed for three benchmark models at various speeds and the results were discussed in terms of total resistance, sinkage and trim. The numerical method was first verified with an appropriate method. Following this, the numerical study was extended to cover a wide range of ship speeds for a comprehensive validation purpose.

4.1. Verification Study

The uncertainty assessment was done in spatial and temporal manner using the GCI method. Fine, medium and coarse analysis cases were created by changing the grid size and time step size as recommended in several studies (Celik et al., 2008; Roache, 1998; Xing and Stern, 2010). The numerical uncertainties were calculated at the design Froude number of each ship model. As can be seen from Table 2 and 3, the uncertainty study was conducted with a tree-grid sensitivity. In spatial uncertainty, all three cases were fit for GCI method, however, KCS and DTC models showed oscillating convergence. In this manner, a finer mesh was adopted and the uncertainty value was calculated using an analysis set of four cases. Also, the temporal uncertainty for the KCS model was calculated with a two-grid sensitivity by using the fine and medium cases since the convergence condition (R) is higher than one (Roache, 1997). The spatial uncertainty study was carried out for fine, medium and coarse grid sizes at a fixed time step while the temporal uncertainty was calculated for fine, medium and coarse time step sizes at a fixed grid size (Eça et al., 2019).

Table 2. Verification study for spatial uncertainty.

Parameter	R_T (KCS)	R_T (DTC)	R_T (ONRT)
N_1	921097	1106898	1907969
N_2	527246	646240	1465159
N_3	348191	429861	973437
φ_1	85.8018	33.3128	3.6095
φ_2	85.3419	33.0741	3.6261
φ_3	87.9133	35.0722	3.8981
R	-0.1788	-0.1194	0.061
U_G (%)	1.230	2.640	0.038

According to the spatial uncertainty, the numerical approach has an oscillating convergence (Table 2) for three ship models since the convergence condition (R) is lower than zero (Stern et al., 2001). For oscillating convergence, the uncertainty values were calculated as described in the related references. The temporal uncertainty has a monotonic convergence trend excluding the KCS case as shown in Table 3. A two-grid GCI approach was employed in this case. In Table 2 and 3, N_i and Δt_i stand for the grid number and time step size for fine, medium and coarse cases. φ belongs to the scaler function for the relevant grid number or time step size. R is the convergence condition. U_G and U_T are the spatial and temporal uncertainty values, respectively.

Table 3. Verification study for temporal uncertainty.

Parameter	R_T (KCS)	R_T (DTC)	R_T (ONRT)
Δt_1	0.040	0.040	0.020
Δt_2	0.056	0.056	0.028
Δt_3	0.080	0.080	0.040
φ_1	85.8018	33.3128	3.6095
φ_2	86.1456	33.5116	3.6164
φ_3	86.4076	33.8449	3.6348
R	1.3122	0.5966	0.3753
U_T (%)	0.0120	1.1033	0.1433

The total uncertainty in the numerical analyses was calculated with the following equation while considering the iterative uncertainty (U_I) as zero (Larsson and Zou, 2014).

$$U_{SN} = \sqrt{{U_G}^2 + {U_T}^2 + {U_I}^2} \tag{4}$$

The total uncertainty was calculated as given in Table 4. The validation uncertainty U_V was not calculated due to the lack of experimental uncertainty data.

Table 4. Total uncertainty values.

Ship	U_{SN} (%)
KCS	1.230
DTC	2.861
ONRT	0.3772

Within the uncertainty assessment, the rest of the analyses for all ship models were conducted using the fine grid and fine time step size.

4.2 Validation Study

Following the verification study, a comprehensive validation study was conducted for all ship models by comparing the numerical results with the available experimental data. In the given figures, URANS stands for the present results. The numerical results were presented through Froude number.

$$Fr = \frac{V}{\sqrt{gL}} \tag{5}$$

Here, V is the ship velocity (m/s) while g is the gravitational acceleration (m/s^2) and L is the ship length (m).

The numerical results of KCS in the model scale were presented in Figure 3-4. Other results given in the figures are the experimental ones of several towing tanks located around the world (Japan, Korea, Denmark and USA). One may see that the numerical values were mostly satisfying when compared with various experimental results (Hino et al., 2021; S. Van et al., 2011) in terms of total resistance, sinkage and trim. Here, the negative value of the sinkage means the ship goes downward while the negative trim value means that the ship is trimmed by the stern. Here, $\sigma$is the calculated or measured sinkage value and it is non-dimensionalized by the length of perpendiculars (L_{PP}).

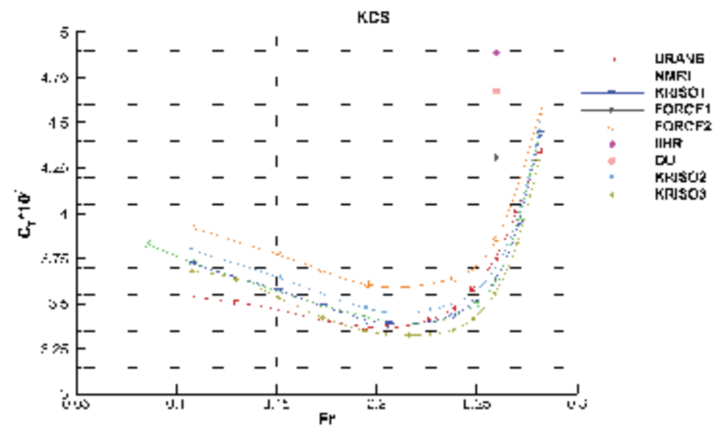


Figure 3. Comparison of total resistance coefficients for KCS.

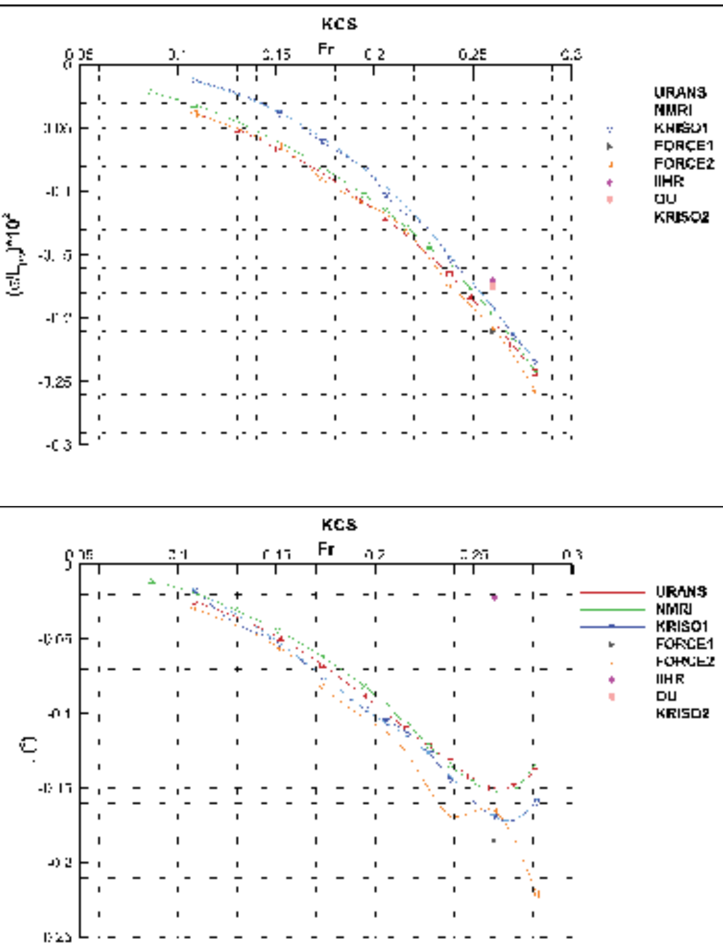


Figure 4. Comparison of sinkage (top) and trim (bottom) values for KCS.

ISSN: 1300-1973, e-ISSN: 2651-530X, Dergi ana sayfası: http://www.gmoshipmar.org/

Figure 5 shows the non-dimensional resistance coefficients of the resistance components. The decomposition of the total resistance was performed as described in the literature (Bertram, 2014). The decomposition of total resistance is given in Equation (6). Here, a comparison could be done through the frictional resistance coefficient and it is found that the numerical trend is similar to the one calculated with the ITTC friction line (ITTC, 1957). The average absolute error is about 2.8%. One may see that the residual resistance coefficient based on the pressure forces start to increase after Fr=0.2 that means the pressure forces start to be dominant.

$$C_T = C_F + C_R \qquad (6)$$

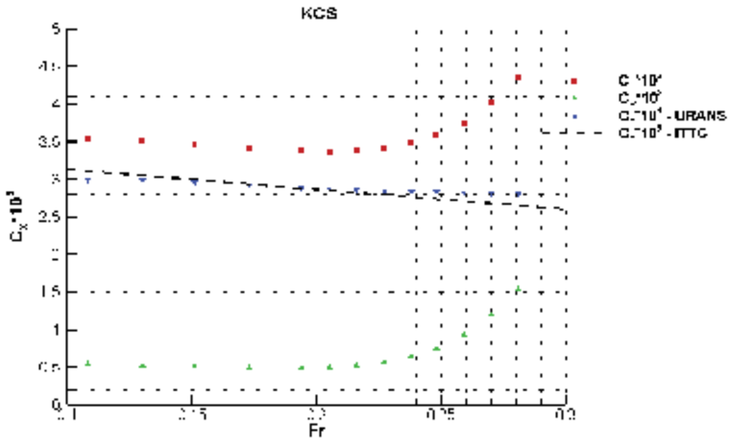
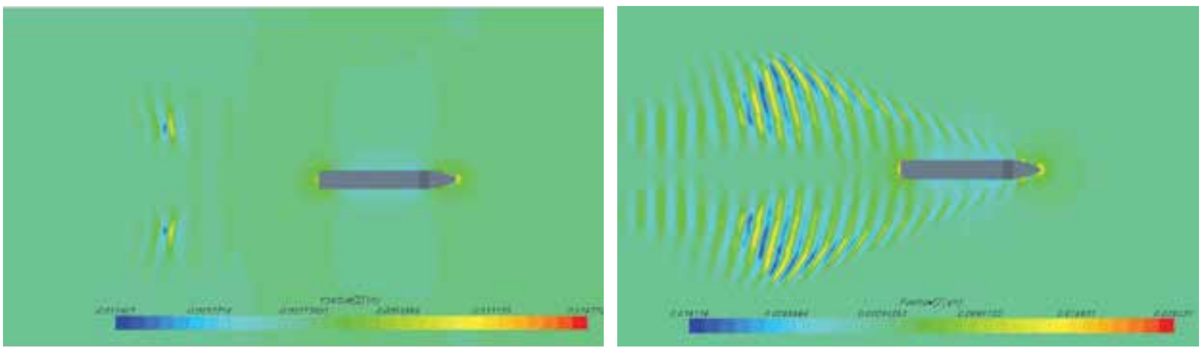
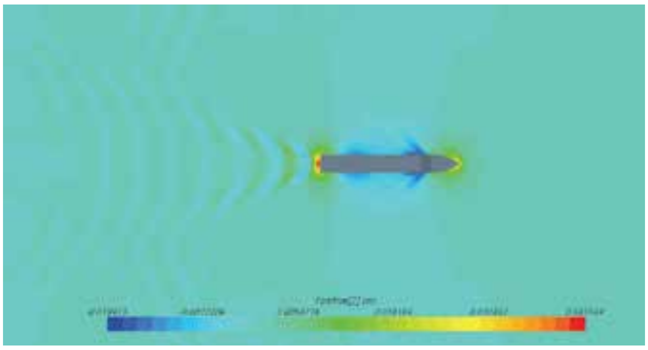


Figure 5. Non-dimensional coefficients of resistance components of KCS.

Figure 6 shows the wave patterns of KCS hull at different Froude numbers. At the lowest one, there is nearly no deformation on the free surface. At Fr=0.151, there are observable deformations while the wave length is too small due to the velocity. Fr=0.205 creates higher waves around the hull while the eave length is larger.



(a) (b)



(c)

Figure 6. Wave patterns around KCS for Fr=0.108 (a), Fr=0.151 (b), Fr=0.205 (c).

The numerical results of DTC in model scale were compared with the experimental results (Moctar et al., 2012) through non-dimensional total resistance coefficient due to the lack of experimental sinkage and trim results. Figure 7 shows that the difference between numerical and experimental results becomes larger with the increase in Froude number. However, the average relative difference is about 3.4%. Figure 8 presents the trim and sinkage values at different Froude numbers.

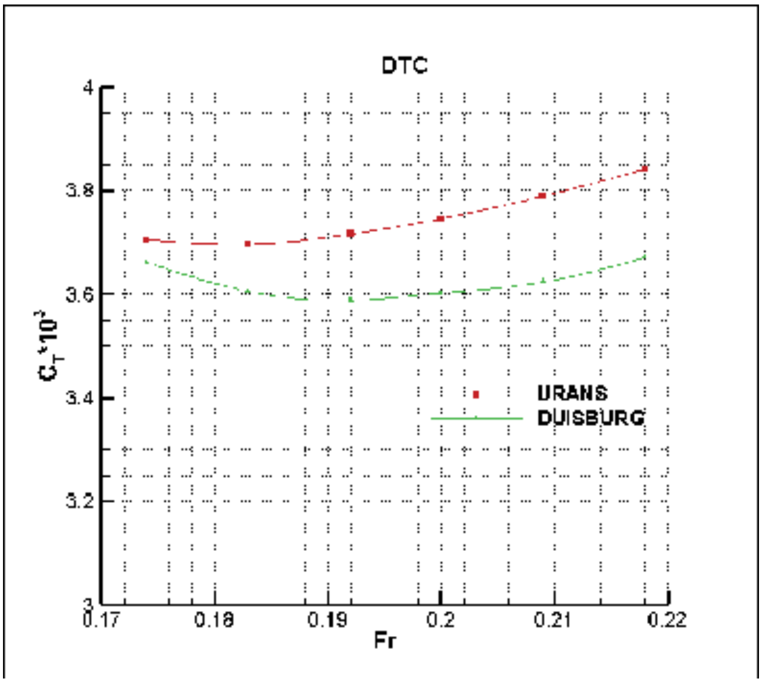


Figure 7. Comparison of total resistance coefficients for DTC.

ISSN: 1300-1973, e-ISSN: 2651-530X, Dergi ana sayfası: http://www.gmoshipmar.org/

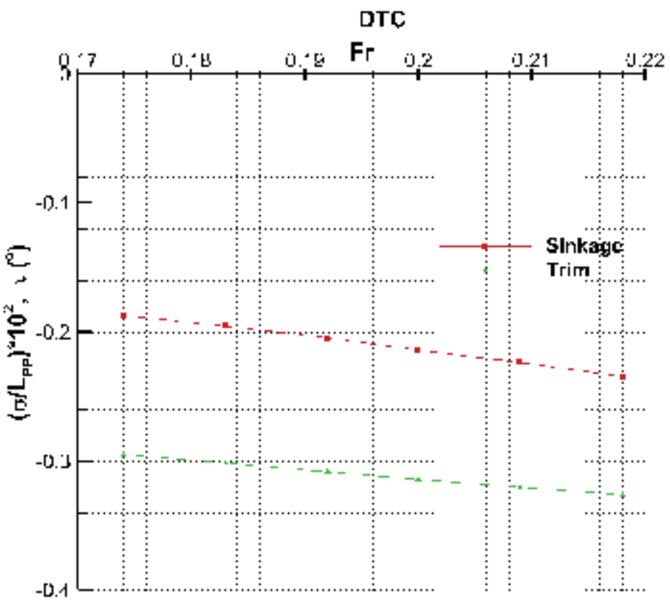


Figure 8. Sinkage and trim values of DTC.

Figure 9 gives the resistance components of DTC hull in the model scale. The frictional resistance coefficient is in good agreement with the ITTC friction line with an average absolute error of 4.2% approximately. It is observed that the CFD method overestimates the frictional resistance coefficient in the whole Froude number range. It may be due to the mesh structure inside the boundary layer, a denser prism layer yielding lower y^+ values may lead to more precise results. Figure 9 also shows that the residual resistance coefficient changes very little with the change in Froude number. That means the effect of the pressure forces changes slightly with the Froude number.

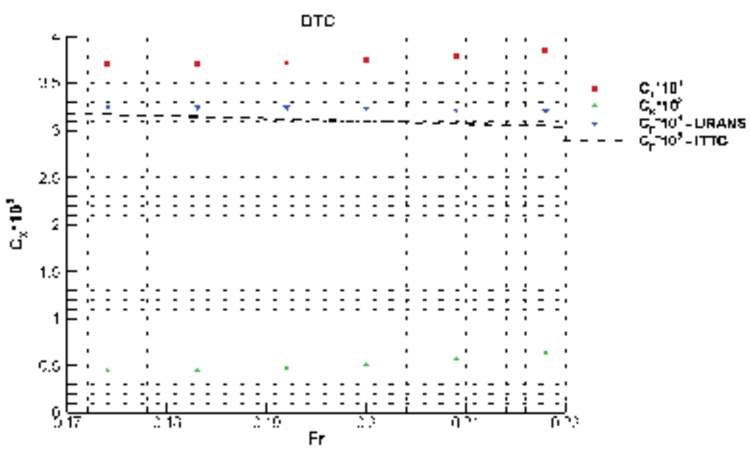
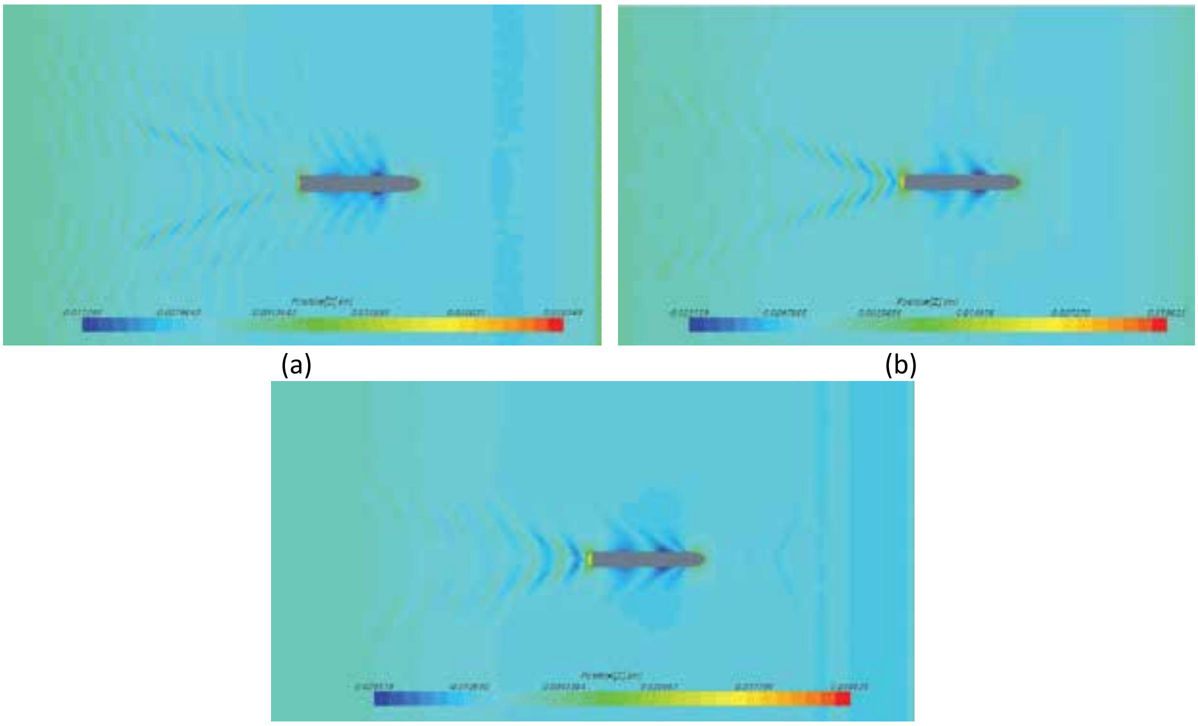


Figure 9. Non-dimensional coefficients of resistance components of DTC.

Figure 10 shows the wave patterns of DTC hull at different Froude numbers. One may see that the wave length increases with the increase in Froude number. The wave deformations are seen prominently.



(a) (b)

(c)

Figure 10. Wave patterns around DTC for Fr=0.174 (a), Fr=0.200 (b), Fr=0.218 (c).

Figure 11 and 12 show the comparison of the present results with other experimental and numerical results available in the literature (Cook, 2011; C. Delen and Bal, 2019) for ONRT. The numerical results show a good match, especially with the model experiments. The present results cover different Froude numbers in terms of total resistance coefficient, sinkage and trim. The relative difference for sinkage and trim becomes higher, however, the results show a similar trend with the other results.

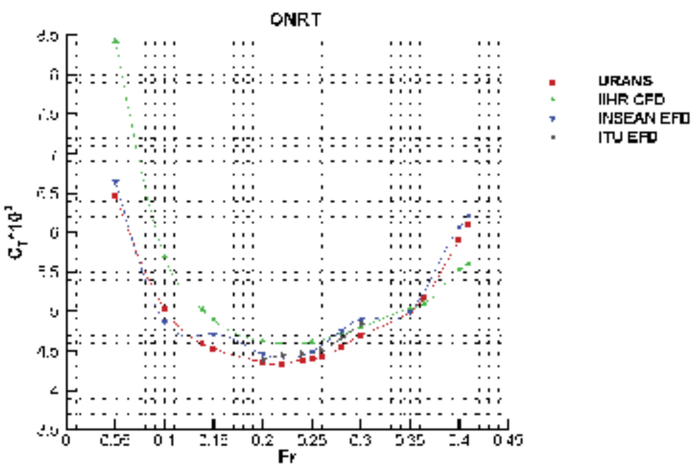


Figure11. Comparison of total resistance for ONRT.

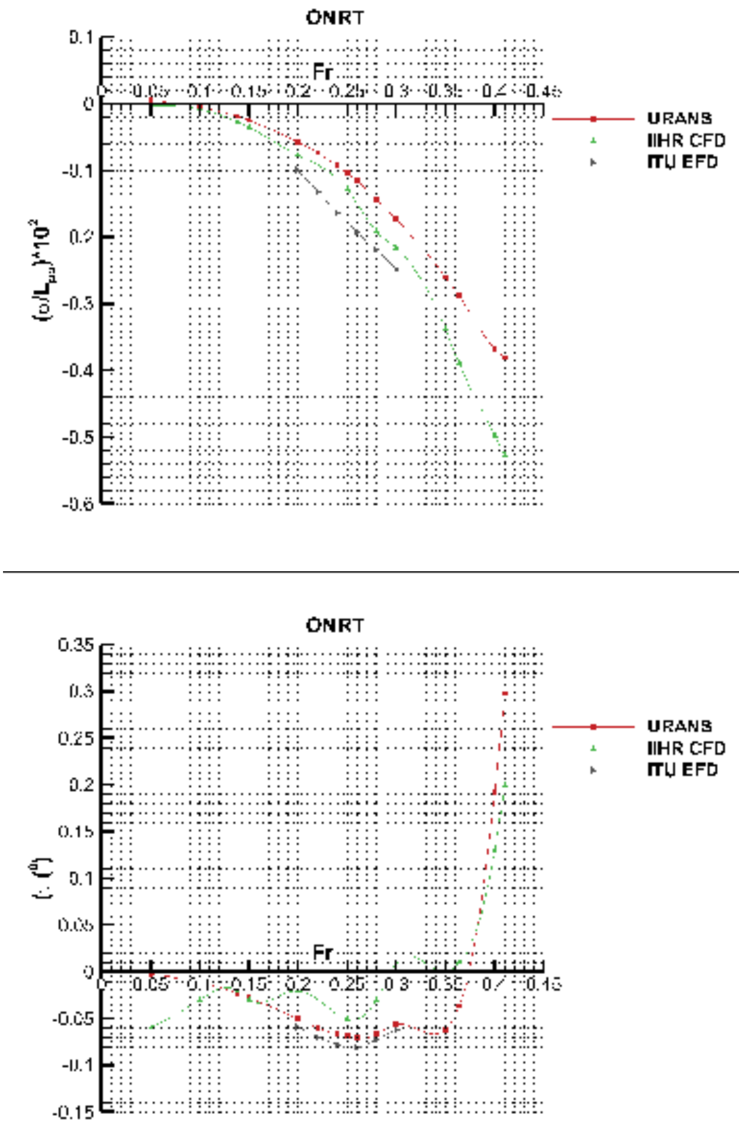


Figure 12. Comparison of sinkage (top) and trim (bottom) values for ONRT.

One may see the non-dimensional resistance coefficients of ONRT bare hull in Figure 13. The average absolute error between the numerical and empirical (ITTC) friction lines is about 2.9%. In addition to this, the trend of the residual resistance coefficient shows that the contribution of the pressure forces decreases with the Froude number until Fr=0.2. After, the trend shows a dramatic increase and finally the pressure forces become as dominant as the shear forces.

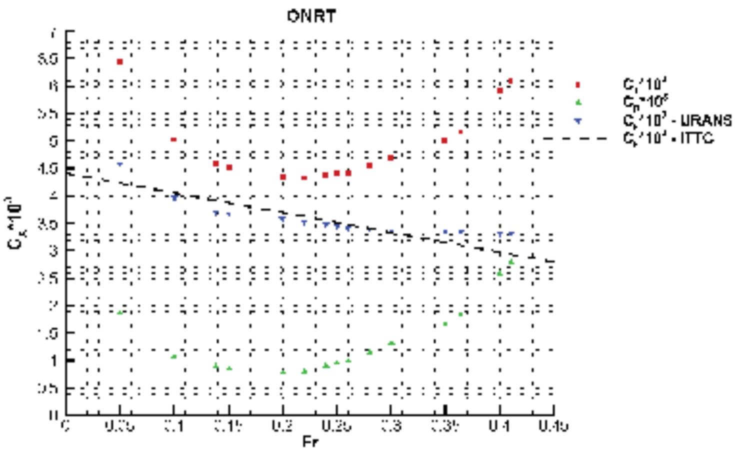
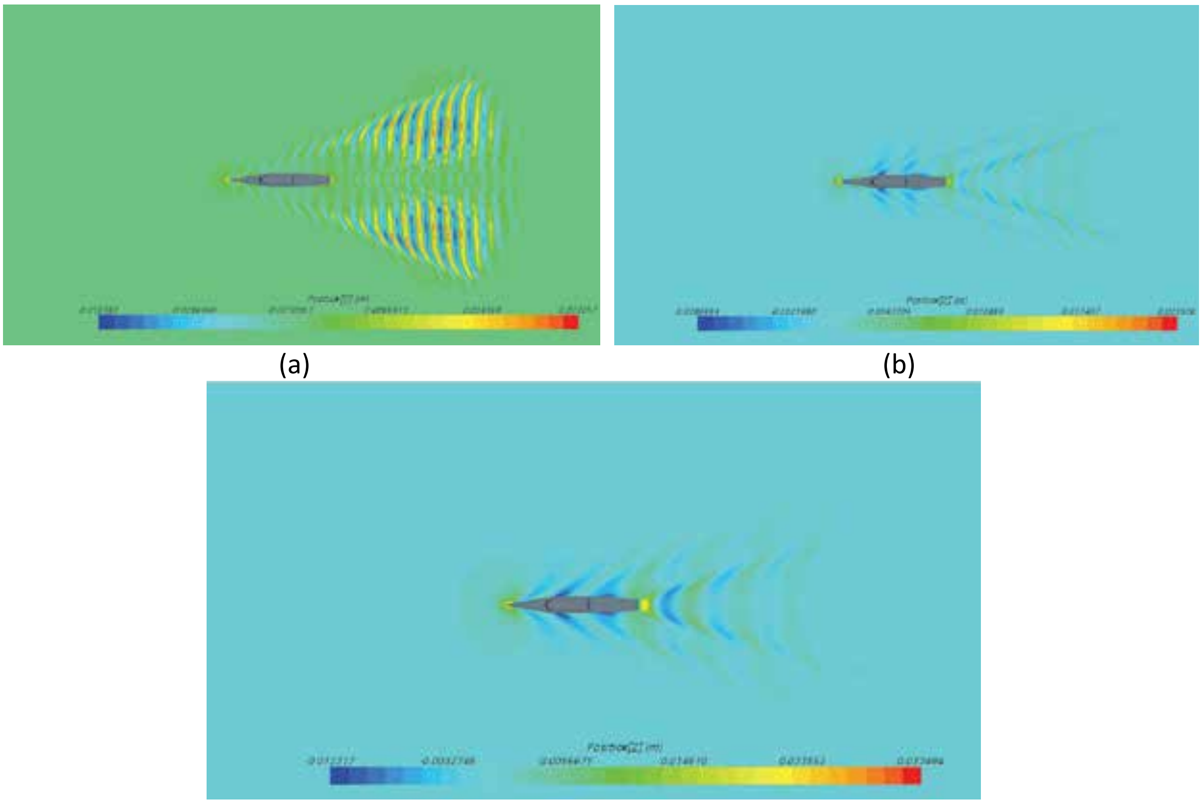


Figure 13. Non-dimensional coefficients of resistance components of ONRT.

Figure 14 shows the wave patterns of ONRT hull at different Froude numbers. Again, there is a correlation between the wave length and Froude number. However, the wave height decreases at Fr=0.200 that is also observed in Figure 13 through the residual resistance coefficient. At Fr=0.250, the deformations become higher in accordance with the increase in the residual resistance coefficient.



(a) (b)

(c)

Figure 14. Wave patterns around ONRT for Fr=0.150 (a), Fr=0.200 (b), Fr=0.250 (c).

5. Conclusion

This study focuses on the numerical investigation of the flow around benchmark ship models. It is aimed to present comprehensive numerical results of three well-known benchmark vessels including comparisons with the experimental results in the literature. Verification was done using the GCI method to obtain the spatial and temporal uncertainty values. The numerical uncertainties were below 3%. In especially ONRT case, the uncertainty values were found below 1%. Using the same refinement ratio for both spatial and temporal uncertainty, it is observed that the temporal analysis sets were more convenient for the GCI method except the KCS model.

A comprehensive validation study was conducted following the verification study. Validation for KCS was done through total resistance coefficient, sinkage and trim values. The results were in good agreement with the experiments especially after Fr=0.2, while for low Froude numbers there is a slight difference in total resistance coefficient. For trim and sinkage, the results were close to the experimental ones in the whole Froude number range. When it comes to DTC model, the validation was done only for the total resistance coefficient due to the lack of experimental data of sinkage and trim. The results showed deviance with the increase in Froude number. This might be caused by the difference in the residual resistance component since the frictional resistance follows the ITTC friction line. ONRT bare hull was validated with various experimental and numerical studies available in the literature. The total resistance coefficient results were in a similar trend with the experimental ones in the whole speed range while the other numerical study was quite far from the experiments in low Froude numbers. In addition to this, the trim values were found close to the experiments while the sinkage values followed a similar trend with the other numerical study.

It is concluded that the numerical method in the present study gives promising results with reasonable grid numbers and time step sizes with acceptable orders of uncertainty. The present study can be extended with the self-propulsion analyses and the validation can be done in terms of self-propulsion point and other propulsive parameters.

6. References

Bertram, V. (2014). *Practical Ship Hydrodynamics* (2nd edition). Elsevier Science.

Can, U., Delen, C., Bal, S. (2020). "Effective wake estimation of KCS hull at full-scale by GEOSIM method based on CFD". Ocean Engineering, 218, 108052. doi:10.1016/j.oceaneng.2020.108052.

Carrica, P. M., Fu, H., Stern, F. (2011). "Computations of self-propulsion free to sink and trim and of motions in head waves of the KRISO Container Ship (KCS) model". Applied Ocean Research, 33(4), 309–320. doi:10.1016/j.apor.2011.07.003.

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Raad, P. E. (2008). "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications". Journal of Fluids Engineering, 130(7), 078001. doi:10.1115/1.2960953.

Cook, S. S. (2011). Effects of headwinds on towing tank resistance and PMM tests for ONR Tumblehome [MSc Thesis, University of Iowa]. Available from Iowa Research Online. doi:10.17077/etd.9t68ik1e.

Cosner, R., Oberkampf, B., Rumsey, C., Rahaim, C., Shih, T. (2006). "AIAA Committee on Standards for Computational Fluid Dynamics: Status and Plans". In 44th AIAA Aerospace Sciences Meeting and Exhibit. American Institute of Aeronautics and Astronautics. doi:10.2514/6.2006-889.

Delen, C., Bal, S. (2019, September 9). "Uncertainty analysis of numerical and experimental resistance tests for ONR Tumblehome". International Maritime Association of the Mediterranean IMAM 2019, Varna, Bulgaria.

Delen, C., Can, U., Bal, S. (2020). "Prediction of Resistance and Self-Propulsion Characteristics of a Full-Scale Naval Ship by CFD-Based GEOSIM Method". Journal of Ship Research, 16. doi:10.5957/JOSR.03200022.

Dogrul, A., Song, S., Demirel, Y. K. (2020). "Scale effect on ship resistance components and form factor". Ocean Engineering, 209, 107428. doi:10.1016/j.oceaneng.2020.107428.

Eça, L., Vaz, G., Toxopeus, S. L., Hoekstra, M. (2019). "Numerical Errors in Unsteady Flow Simulations". Journal of Verification, Validation and Uncertainty Quantification, 4(021001). doi:10.1115/1.4043975.

Ferziger, J. H., Peric, M., Robert, L. (2020). Computational Methods for Fluid Dynamics (4th Edition). Springer International Publishing.

Guo, H., Zou, Z., Liu, Y., Wang, F. (2018). "Investigation on hull-propeller-rudder interaction by RANS simulation of captive model tests for a twin-screw ship". Ocean Engineering, 162, 259–273. doi:10.1016/j.oceaneng.2018.05.035.

Hino, T., Stern, F., Larsson, L., Visonneau, M., Hirata, N., Kim, J. (Eds.). (2021). Numerical Ship Hydrodynamics: An Assessment of the Tokyo 2015 Workshop. Springer International Publishing. doi:10.1007/978-3-030-47572-7.

ITTC. (1957). Report of Resistance Committee. Proceedings of 8th ITTC, Madrid, Spain.

ITTC. (2014a). 7.5-03-02-03 Practical Guidelines for Ship CFD Applications. In ITTC - Recommended Procedures and Guidelines.

ITTC. (2014b). 7.5-03-01-01 Uncertainty Analysis in CFD, Verification and Validation Methodology and Procedures. In ITTC - Recommended Procedures and Guidelines.

Kahramanoglu, E., Cakici, F., Dogrul, A. (2020). "Numerical Prediction of the Vertical Responses of Planing Hulls in Regular Head Waves". Journal of Marine Science and Engineering, 8(6), 455. doi:10.3390/jmse8060455.

Kim, W. J., Van, S. H., Kim, D. H. (2001). "Measurement of flows around modern commercial ship models". Experiments in Fluids, 31(5), 567–578. doi:10.1007/s003480100332.

Kinaci, O. K., Gokce, M. K., Delen, C. (2020). "Resistance experiments and self-propulsion estimations of Duisburg Test Case at 1/100 scale". Ship Technology Research, 67(2), 109–120. doi:10.1080/09377255.2020.1729454.

Kok, Z., Duffy, J., Chai, S., Jin, Y., Javanmardi, M. (2020). "Numerical investigation of scale effect in self-propelled container ship squat". Applied Ocean Research, 99, 102143. doi:10.1016/j.apor.2020.102143.

Larsson, L., Zou, L. (2014). "Evaluation of Resistance, Sinkage and Trim, Self Propulsion and Wave Pattern Predictions". In L. Larsson, F. Stern, & M. Visonneau (Eds.), Numerical Ship Hydrodynamics (pp. 17–64). Springer Netherlands. doi:10.1007/978-94-007-7189-5_2.

Menter, F. R. (1994). "Two-equation eddy-viscosity turbulence models for engineering applications". AIAA Journal, 32(8), 1598–1605. doi:10.2514/3.12149.

Menter, F. R. (2009). "Review of the shear-stress transport turbulence model experience from an industrial perspective". International Journal of Computational Fluid Dynamics, 23(4), 305–316. doi:10.1080/10618560902773387.

Moctar, O. el, Shigunov, V., Zorn, T. (2012). "Duisburg Test Case: Post-Panamax Container Ship for Benchmarking". Ship Technology Research, 59(3), 50–64. doi:10.1179/str.2012.59.3.004.

Ozdemir, Y. H., Cosgun, T., Dogrul, A., Barlas, B. (2016). "A Numerical Application to Predict The Resistance and Wave Pattern of KRISO Container Ship". Brodogradnja : Teorija i Praksa Brodogradnje i Pomorske Tehnike, 67(2), 47–65. doi:10.21278/brod67204.

Pereira, F. S., Eça, L., Vaz, G. (2017). "Verification and Validation exercises for the flow around the KVLCC2 tanker at model and full-scale Reynolds numbers". Ocean Engineering, 129, 133–148. doi:10.1016/j.oceaneng.2016.11.005.

Richardson, L. F. (1911). "The Approximate Arithmetical Solution by Finite Differences of Physical Problems Involving Differential Equations, with an Application to the Stresses in a Masonry Dam". Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character, 210, 307–357.

Roache, P. J. (1998). "Verification of Codes and Calculations". AIAA Journal, 36(5), 696–702. doi:10.2514/2.457.

Roache, P. J. (1997). "Quantification of uncertainty in computational fluid dynamics". Annual Review of Fluid Mechanics, 29, 123–160. Scopus. doi:10.1146/annurev.fluid.29.1.123.

Sezen, S., Dogrul, A., Delen, C., Bal, S. (2018). "Investigation of self-propulsion of DARPA Suboff by RANS method". Ocean Engineering, 150, 258–271. doi:10.1016/j.oceaneng.2017.12.051.

Shen, Z., Wan, D., Carrica, P. M. (2015). "Dynamic overset grids in OpenFOAM with application to KCS self-propulsion and maneuvering. Ocean Engineering, 108, 287–306. doi:10.1016/j.oceaneng.2015.07.035.

Stern, F., Wilson, R. V., Coleman, H. W., Paterson, E. G. (2001). Comprehensive Approach to Verification and Validation of CFD Simulations—Part 1: Methodology and Procedures". Journal of Fluids Engineering, 123(4), 793–802. doi:10.1115/1.1412235.

Terziev, M., Tezdogan, T., Oguz, E., Gourlay, T., Demirel, Y. K., Incecik, A. (2018). "Numerical investigation of the behaviour and performance of ships advancing through restricted shallow waters". Journal of Fluids and Structures, 76, 185–215. doi:10.1016/j.jfluidstructs.2017.10.003.

Tezdogan, T., Demirel, Y. K., Kellett, P., Khorasanchi, M., Incecik, A., Turan, O. (2015). "Full-scale unsteady RANS CFD simulations of ship behaviour and performance in head seas due to slow steaming". Ocean Engineering, 97, 186–206. doi:10.1016/j.oceaneng.2015.01.011.

Tezdogan, T., Incecik, A., Turan, O. (2016). "A numerical investigation of the squat and resistance of ships advancing through a canal using CFD". Journal of Marine Science and Technology, 1–16. doi:10.1007/s00773-015-0334-1.

Van, S., Ahn, H., Lee, Y., Kim, C., Hwang, S., Kim, J., Kim, K., Park, I. (2011). "Resistance characteristics and form factor evaluation for geosim models of KVLCC2 and KCS". Proceeding of 2nd International Conference on Advanced Model Measurement Technology for EU Maritime Industry, 282–293.

Van, S. H., Kim, W. J., Yoon, H. S., Lee, Y. Y., Park, I. R. (2006). "Flow measurement around a model ship with propeller and rudder". Experiments in Fluids, 40(4), 533–545. doi:10.1007/s00348-005-0093-6.

Wilcox, D. C. (2008). "Formulation of the k-w Turbulence Model Revisited". AIAA Journal, 46(11), 2823–

2838. doi:10.2514/1.36541.

Xing, T., Stern, F. (2010). "Factors of Safety for Richardson Extrapolation". Journal of Fluids Engineering, 132(061403). doi:10.1115/1.4001771.

Zhang, Z. (2010). "Verification and validation for RANS simulation of KCS container ship without/with propeller". Journal of Hydrodynamics, Ser. B, 22(5, Supplement 1), 932–939. doi:10.1016/S1001-6058(10)60055-8.