ISSN (e): 2250 – 3005 || Volume, 08 || Issue, 12|| December – 2018 ||
International Journal of Computational Engineering Research (IJCER)

A review on the advancement of CFD technique in ship hydrodynamics

Abhiroop K[1] , Varada Saidas[1], Shameem B.M[2]
[1]Undergraduate Student, Department of Naval Architecture and Offshore Engineering, AMET Deemed to be University, Chennai
[2]Assistant Professor, Department of Naval Architecture and Offshore Engineering, AMET Deemed to be University, Chennai
Corresponding Author: Abhiroop K

ABSTRACT: *Computational Fluid Dynamics (CFD) is one of the significant methods used to model and analyse the transport phenomenon of fluids. It has been widely used in many engineering fields to understand the flow pattern and the dynamic effects due to fluid motions. The developments of CFD in the field of marine technology, especially in ship hydrodynamics for last two decades is exceptional. Besides the recent advancements in computational capability in saving the computational time, the graphical user interface has also become more user friendly. The present paper sets the scene with a review on the applications of CFD in ship hydrodynamics followed by some suggestions for future work. The paper discusses the progress in estimation of resistance, propulsion, interaction studies, motion analysis and maneuvering aspects in ship hydrodynamics.*
Keywords: *Computational Fluid Dynamics (CFD), hydrodynamics, resistance, propulsion, seakeeping, maneuvering*

Date of Submission: 16-01-2019 Date of acceptance: 28-01-2019

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a method of simulation of fluid flow to visualize, predict and evaluate the effect of flow on various engineering applications. Earlier in the 1970s, the computer applications and ability of solving the fluid problems using the computers were challenging due to their processing capabilities. Over the past few decades, the technology has developed with regard to the processing speed as well as the accuracy in the solutions for complex problems including heat transfer, particle tracking, chemical reactions, aerodynamic and marine problems.

Simulation Based Design (SBD) is becoming more popular in marine industry against the traditional build and test approach. CFD as a simulation-based tool is emerging against the traditional approach byreplicating the real scenario of ship dynamics in the computer. Compared to the earlier CFD tools, the latest high fidelity CFD tools has the capability to incorporate wide variety of simulations and at the same time it has become economical against experiments and test facilities. It has developedas an efficient tool to optimize many hydrodynamic problems even from the concept stage to the final design stage. The CFD simulations are contributing in such a way that a much better understanding of the problem is accomplished which sometimes is not possible with the experiments. The modern CFD tools has become portable and has user friendly GUI with wide range of options for variation of the flow as well as other computational parameters.Though CFD has many advantages against the potential flow and the experiments, in many cases especially in marine applications the accuracy of the result are yet to improve and hence needed more validation studies.

Many problems related to hydrodynamics is very difficult to predict using captive facilities or free running tests due to the need for instantaneous visualization or sophisticated measurement tools. The usage of CFD solver for ship hydrodynamics includes estimation of resistance and self-propulsion, characteristics of propulsive devices, maneuvering studies and seakeeping predictions. Fig 1 shows the trends in usage of CFD for different hydrodynamic problems. Resistance prediction is one of the dominant areas of CFD application in Naval Architecture, 64% of resistance estimation is currently using CFD technique (ITTC, 2011). These studies also involve sinkage and trim, boundary layer flow analysis, wake vortices and wave pattern around the ship hull. Most of these cases have been validated with the experiments and standard methodologies are proposed by many researchers. The percentage usage of CFD for various resistance estimation and flow parameters are given in Fig 2 (ITTC 2011). Seakeeping predictions using CFD is also increasing rapidly since Tokyo 2005 CFD workshop by National Maritime Research Institute (NMRI). Progress in computational ship hydrodynamics has been suitably benchmarked by ITTC and also by many CFD workshops.

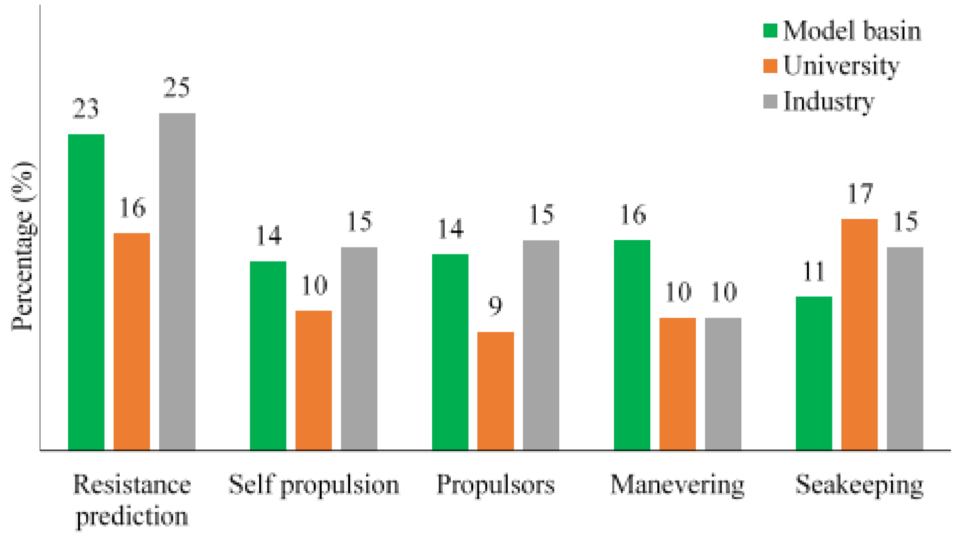

Fig 1: Usage of CFD forhydrodynamic problems in Naval Architecture

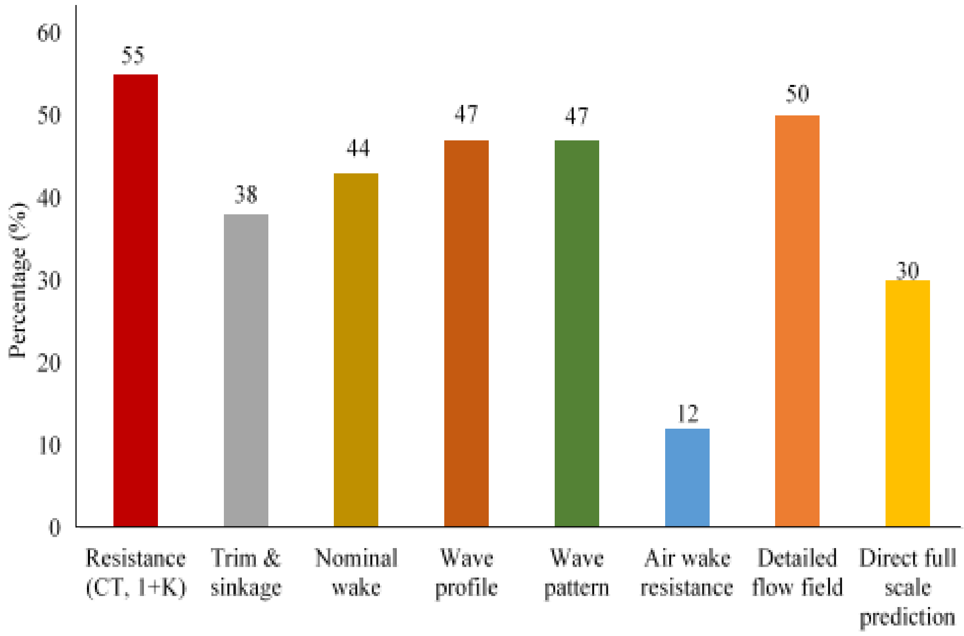

Fig 2: Usage of CFD in resistance and flow simulations

The present paper reviews the various applications of CFD in ship hydrodynamic problems especially in resistance, propulsion, seakeeping and maneuvering.The paper also discusses different strategies adopted for computation and meshing. For a typical ship hydrodynamic problem, the CFD solution is obtained through various stages as depicted in Fig. 3.

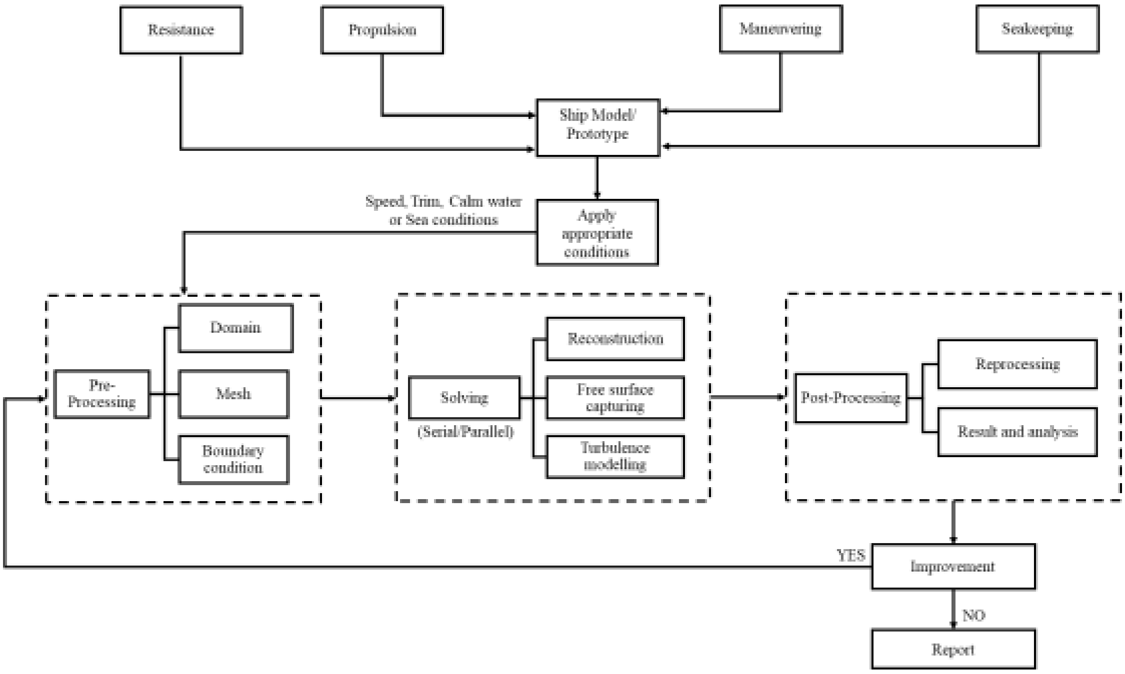

Fig 3: CFD procedure to obtain a typical ship hydrodynamic solution

II. THEORETICAL BASE OF CFD

The governing equations of CFD are the conservation of mass, momentum and energy. Most of the marine related simulations require only mass and momentum conservation, the basic equations are given in Eqn.1 and 2

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\bar{v}) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho\bar{v}) + \nabla.(\rho\bar{v}\bar{v}) = -\nabla p + \nabla.\bar{\bar{\tau}} + \rho g \qquad (2)$$

Where $\bar{v}$, p and $\bar{\bar{\tau}}$ are the velocity vector, static pressure and the stress tensorrespectively. The stress tensor is expressed as,

$$\bar{\bar{\tau}} = \mu\left|\left(\nabla\bar{v}+\nabla\bar{v}^{T}\right) - \frac{2}{3}\nabla.\bar{v}\, I\right| \qquad (3)$$

Where μ is the viscosity term, I the unit tensor, and $\left(\nabla\bar{v}+\nabla\bar{v}^{T}\right)$incorporates the effect of volume dilation. The turbulence is modelled with the Reynolds averaging approach, then the Reynolds Averaged Navier Stokes Equation (RANSE) is written in the following form:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial \rho}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right]+\frac{\partial}{\partial x_j}(-\rho\bar{u}'_i\bar{u}'_j) \qquad (4)$$

where δ_{ij} is the Kronecker delta, and $\frac{\partial}{\partial x_j}(-\rho\bar{u}'_i\bar{u}'_j)$ the Reynolds stresses.

III. COMPUTATIONAL SET-UP FOR SHIP HYDRODYNAMIC PROBLEMS

This section discusses the boundaries, domain set-up and meshing parameters considered for pre-processing of ship hydrodynamic simulations. In CFD, the important stages to obtain the accurate solution is setting of the domain size and the generation of appropriate grid. CFD computational domain for ship hydrodynamic problems in calm water in ship heading condition usually has three fixed boundaries (ITTC, 2014)namely the hull, symmetry plane and the still water surface. Additionally, the domain consists of a closed region around the hull or the vessel with velocity inlet on the upstream and the pressure outlet at the downstream end. To minimize the wall effect on the ship, the domain has to be fixed far from the object. Thesetting of the domain size usually carried out after analysing through domain study to reach an appropriate dimension and hence the effects are minimum or negligible.

Finer the grid size greater the computational capability and hence better the simulation results. Solving the Reynold Averaged Navier Stokes Equation (RANSE) with appropriate turbulence models is the traditional approach in CFD for hydrodynamic problems but recently the researchers are improving the technique to use Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) where greater computing facility is needed. The research is focusing on improving the solving techniques and simultaneously increasing the high-performance computing facilities for full scale simulations [53]. The meshing technique can be broadly classified into structured and unstructured grid. For complex geometries always, unstructured meshing is preferred. More than 90% of the simulations were conducted using grids smaller than 10M points[51]. The different meshing approaches is explained in detail, See [5, 9].

3.1 Estimation of resistance and propulsion characteristics

The domain size for resistance has been standardized by the ITTC 2014 proceedings. A comparison between the ITTC 2011 and 2014 recommendations are given in Table 1. "L" represents the length between perpendicular of the ship and the size is defined from the fore and aft part of the ship.

Table 1: Domain size for resistance estimation using CFD

Sl.N	Boundary	2011 [2]	2014 [16]
1	Upstream -Velocity inlet	1 – 2 L	1 L
2	Downstream - Pressure Outlet	3 – 5 L	2 L
3	Symmetry plane	1 L	1 L
4	Top (above deck)– Wall/ Symmetry	0.5 L	0.5 L
5	Bottom (below keel) - Wall / Symmetry	1L	1L

In most of the literature, for self-propulsion and behind ship condition, same domain size for resistance is used with an additional cylindrical domain at the stern region incorporating the propulsion system of the ship [2, 16].For open water tests, generally in the literature a cylindrical domain is selected for the CFD analysis. A standard domain size for these simulations are not given explicitly in the available literature even though some researchers have come with validation studies where the results are matching with the experiments. The domain size for the open water tests which may be used for CFD simulations are given in Table 2, where D is diameter of the propeller.

Table 2: Domain size for open water characteristics using CFD

Sl.N	Boundary	2010 [37]	2013 [11]	2014 [33]
1	Upstream – Velocity Inlet	2.5 D	3 D	2 D
2	Downstream- Pressure Outlet	3.5 D	4 D	4 D
3	Radial direction - Wall	2 D	4 D	4 D

The progress in the prediction of resistance and propulsion has become consistent nowadays due to the advancement of high computing facilities with parallel computing. This would help the user to define very fine grids around the hull and propeller and hence better results by an order of magnitude than a decade ago. In 1990s, the literature shows there is a gradual increase in the usage of total grid size from thousands to only lakhs [84]. In last two decades it has been observed that there is a drastic development, from millions to tens of millions [29,52,27,82]. Recently the simulation of hull with rotating propeller and rudder become more feasible due to the development of dynamic meshing strategy such as sliding or overset grid [36].The overset grid helps to model simultaneously the multibody applications such as sinkage and trim, propeller rotation, vessel and rudder movement. Furthermore, overset grids provide a convenient means of locally refining grids by being able to embed blocks of finer grids (ITTC, 2011). The various applications of overset grid would be discussed in the following sections.

3.3 Estimation of seakeeping and maneuvering performance

ITTC practical guidelines for ship CFD applications (ITTC, 2011) suggests for ship seakeeping analysis in waves, the inlet boundary should be at a distance of 1 to 2 ship lengths away from the vessel, whereas the outlet boundary should be located 3 to 5 ship lengths downstream in the domain. The literature shows the domain for head sea conditions have been studied with symmetry plane and also with the full domain. Other wave conditions such as beam, quartering and following seas and the dynamic effects of the ship with respect to seakeeping are still in a developing state. The domain size for seakeeping studies depends upon the type of waves such as short or long waves [14]. It is calculated on how many wave cycles can advance in a domain, for example 8 cycles for short waves and 2 cycles for long waves. Velocity inlet boundary condition was applied at the upstream of the fluid domain where incident regular waves were generated. Usually a wave damping zone would be generated within the domain to eliminate the reflections of waves from the boundaries. The top, bottom and side walls of the domain were all selected as velocity inlets, which avoids velocity gradient occurring from the boundaries as in the use of a slip-wall condition. The available domain data used for seakeeping analysis in head sea condition is tabulated in Table 3.

Table 3: Domain size for seakeeping in head sea condition

Sl.N	Boundary	Domain size				
		2011 [2]	2014 [36]	2016 [45]	2016 [86]	2017 [14]
1	Fore stream – Velocity Inlet	1 – 2 L	1.0 L	1.6 L	1.0 L	1.6 L
2	Downstream – Pressure Outlet	3 – 5 L	2.0 L	5.1 L	3.5 L	1.6 L
3	Starboard	NA*	1.0 L	1.0 L	1.5 L	1.0 L
4	Port	NA*	1.0 L	1.0 L	1.5 L	1.0 L
5	Top (above draft)	NA*	1.0 L	1.0 L	0.25 L	NA*
6	Bottom (below model keel)	NA*	1.0 L	2.0 L	1.0 L	1.0 L

*NA – Not Available

The estimation of hydrodynamic derivatives for maneuvering using PMM tests were carried out by various researchers with reasonable validation in calm waters. The domain size could be standardized with these simulations. See Table 4 for a comparison of domain set-up considering five literatures.It is always difficult to simulate the real condition of propeller rudder interaction effects for maneuvering studies to estimate the turning parameters. The simulation of turning circle is reported in many literatures recently though the actual scenario is yet to be modelled and validated. With the advancement of computational architecture, this may be possible in near future.

Table 4: Domain size for hydrodynamic derivatives using PMM tests

Sl.N	Boundary	Domain size				
		2016 [23]	2016 [32]	2015 [18]	2011 [2]	2016 [23]
1	Fore stream – Velocity Inlet	1	1	1	1	2
2	Downstream – Pressure Outlet	4	2	2	2	4
3	Starboard	b*	2	1.5	1.5	3
4	Port	b*	2	1.5	1.5	3
5	Bottom	10d	20d	1.44	1	1

*The distances b from the hull to the starboard side is set as b/L = 1.2,0.5,0.35,0.25

To obtain a good solution and control over the simulation of wide range of seakeeping and maneuvering analysis, dynamic overset meshing is recommended [45]. This grid method would allow the ship to move independently in the computational domain and at the same time the propeller would be rotating around the rotating axis of the propeller [36]. The traditional approach of meshing for the dynamic simulations of seakeeping is of great challenge, since it is difficult to simulate the real scenario of self-propulsion in waves considering the motions and the rotation of propeller and rudder position. Dynamic overset meshing provides an effective way to overcome these challenges. See Fig.4 for the overset region creation for self-propulsion case in waves. Fig 5 depicts the grid distribution of overset for simulating the turning circle for estimating the turning parameters.

As discussed in Section 3.1, a study has been carried out to understand the trend of grid evolution over last three decades involving the ship hydrodynamic problems. Fig 6 shows the evolution of grids in ship hydrodynamics. As mentioned earlier, it is observed from the graph that due to the development in parallel computing along with the high computational power, after 2008 the order of magnitude of grid size has increased by tens of millions.

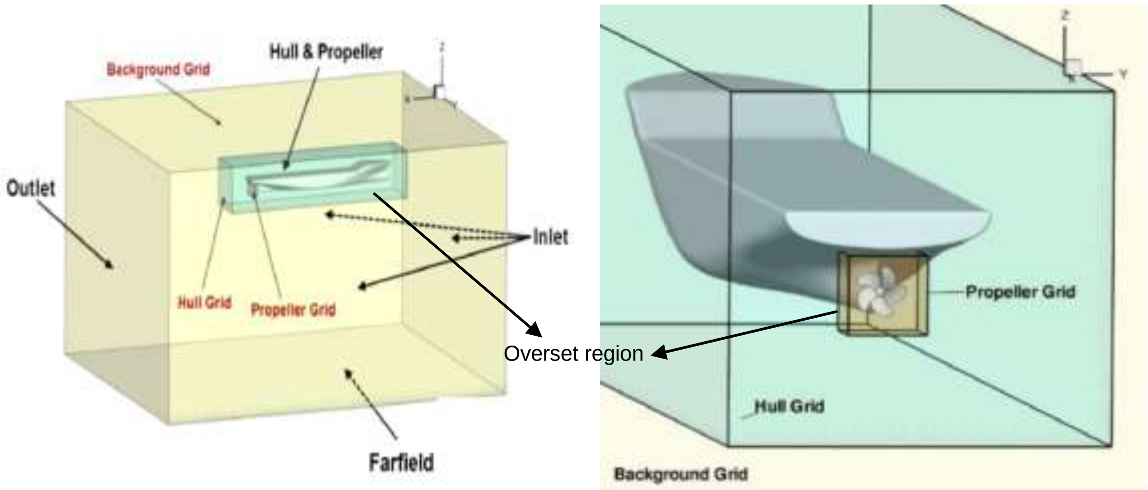

Fig 4: Dynamic meshing strategy applied for ship propulsion[20]

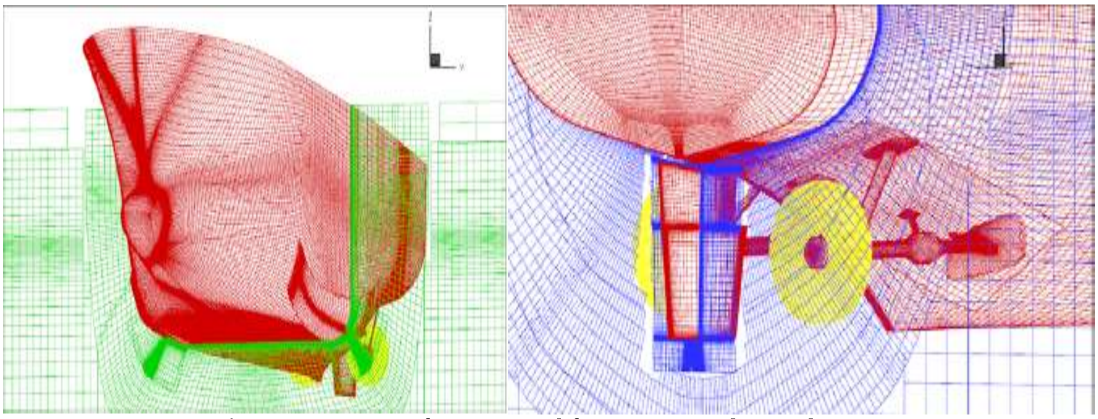

Fig 5: Generation of overset grid for turning circle simulations

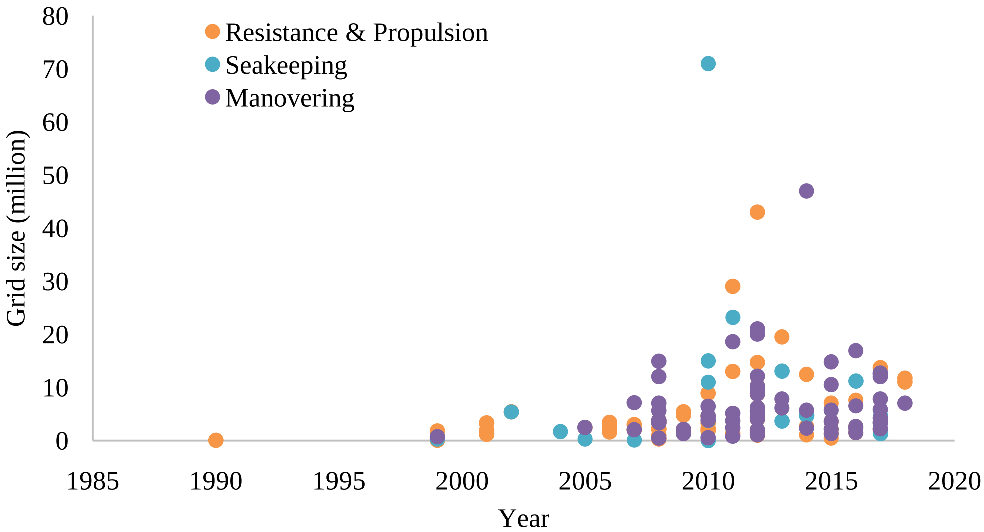

Fig 6: Evolution of grid size for ship hydrodynamic CFD simulations

IV. STATE OF ART - CFD IN SHIP HYDRODYNAMICS

Earlier in 1990s CFD for marine applications was in initial stage and the researchers were using it as a tool to assist the established design [22]. After 21st ITTCrecommendations in 1996, many works related to resistance and propulsion has been reported in the literature. It is observed that in this period of development,the CFD results are not completely reliable for establishing the accurate flow information due to poor capability of simulating the actual turbulence scenario around the stem and stern of the ship with propeller and rudder. But it is reported that for the estimation of resistance alone, the selection of turbulence models has less significance [1, 35]. The literature shows progressive development in numerical simulations with encouraging results especially for resistance and propulsion which may replace Experimental Fluid Dynamics (EFD) in near future [1, 4]. Nowadays, CFD is extensively used alone or in combination for various hydrodynamic problems in marine industry. This section outlines the recent progress that have been observed in each of the traditional naval architecture areas viz., resistance and propulsion, seakeeping and maneuvering.

4.1 Resistance and Propulsion

CFD has been extensively used to predict the wave pattern and the vessel performance in calm water with improved visualization techniques for velocity and pressure distribution around the hull. This detailed information of near and far flow field along with the capability to handle arbitrary geometries would significantly help in optimizing the hull for an efficient or innovative design. The literature also discusses the effect of drag, sinkage, trim, propeller characteristics and propeller-hull interaction in deep as well as shallow water for a range of small to large Froude number [4]. The present paper restricting the discussion to only deep-sea conditions.

For resistance and propulsion, the modelling approach mostly used is RANS and Large Eddy Simulation (LES), and the surface capturing is carried out using Volume of Fluid (VOF) method [1,70]. The turbulence model mostly found in the literature is k-ω model [4].Many investigations have reported on the simulation and validation studies of the propeller, rudder and hull interaction effects with good accuracy[47,56,68]. More review on similar kind of studies is reported in [4]. Recent literature shows the dynamic overset mesh has more control over the traditional meshing approach [63, 47, 61, 65]. Shen et al (2015) compared the overset and non-overset grid approach for propeller characteristics and self-propulsion studies. It is concluded that K_T , K_Q and η are well predicted by the dynamic overset grid approach . The results obtained from this analysis for open water test is shown in Fig 7. CFD analysis of broaching for a model surface combatant with explicit simulation of moving rudders and rotating propellers were carried out using overset method by Carrica et al. The overset strategy applied for simulating the rotating propeller andmoving rudder in broaching condition for vessel is shown in Fig 8.

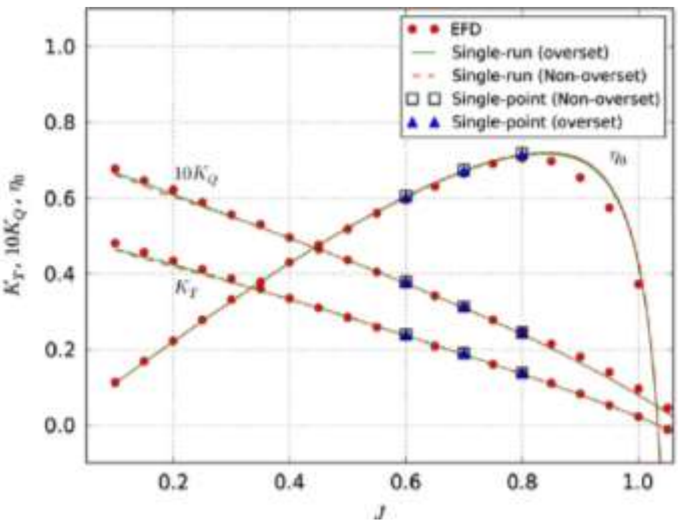

Fig 7: Comparison of overset with non-overset grid approach [15]

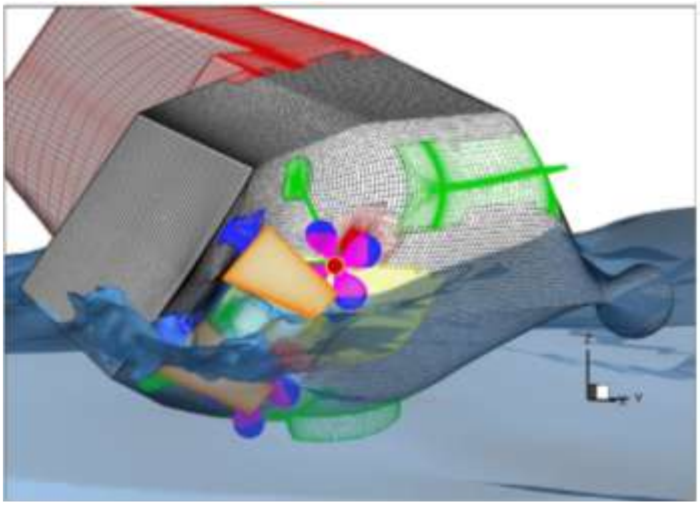

Fig 8: Application of overset strategy for rotating propeller and moving rudder [65]

Earlier it was difficult to generate complex dynamics of propellers behind the ship to predict the underwater noise and its impact on marine ecosystem. Nowadays, it is possible to carry out numerical simulations to study the effects of underwater noise from the moving ship and rotating propeller with reasonable accuracy [20, 21]. Recently, it is reported the thruster hull interaction effects for a drillship with 6 azimuth thrusters, three at aft and three at forward, using dynamic meshing and validated with experimental results for thruster hull interaction coefficients [47]. The adopted meshing strategy is depicted in Fig 9 and 10 respectively. It is concluded that the CFD can be a cost-efficient and accurate method to determine the thruster-hull interaction effects at bollard pull conditions for a typical offshore vessel.

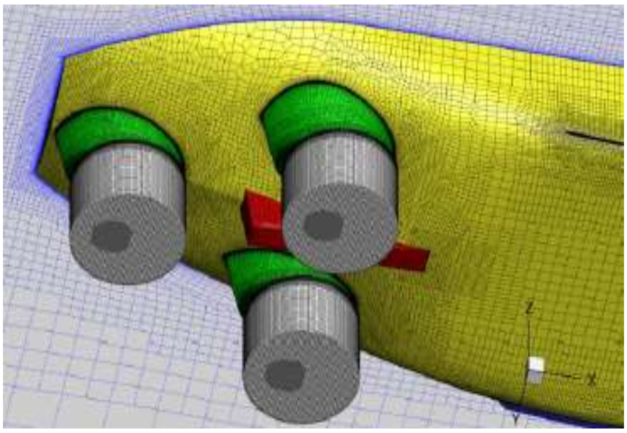

Fig 9: Mesh around the aft of drill ship for 3 azmuthing propeller

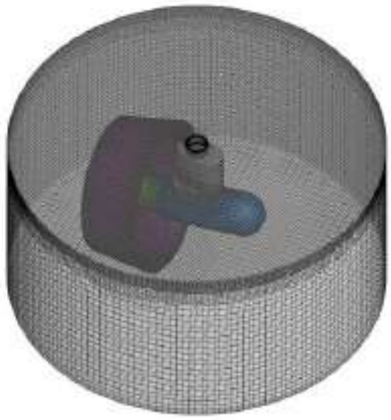

Fig 10: Cylindrical sub domain around the propeller for dynamics

The advancement of CFD simulations has made it possible to estimate the drag with reasonable accuracy on the hull considering the effect of fouling[30, 13, 44]. For these kind of simulations, total number of cells needed would be high and also the time step selected would be very small.This kind of mesh density is required to capture the minute details of fouling on the hull surface. And hence the representation of mesh size should be in micrometers with high computing capability.Nowadays, CFD is capable of simulatingmulti-phase flow conditions with much accuracy such as the full scenario of ship sailing considering both the water and wind effects.The effect of wind on total resistance is predicted and the error deviation with EFD is compared and the results are within 5% [15]. The wind effect on the superstructure and comparison of such results is shown in Fig 11 and 12 respectively.

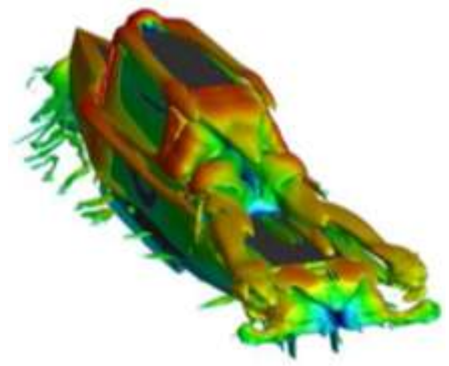

Fig11: Simulation of wind effect over superstructure

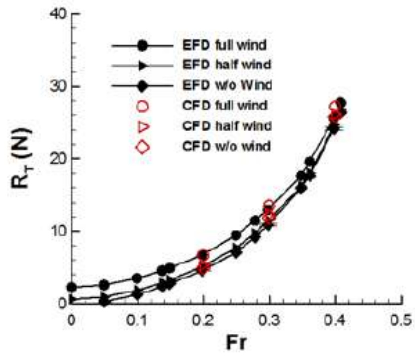

Fig 12: Comparison of CFD with EFD for total resistance

A comparison of the average error percentage on the results for the resistance and propulsion using CFD is plotted, See Fig 13. From the review of literature, it has been observed that sufficient literature on validation of CFD with experiments were reported in between 2004 to 2018. For this period, the authors are able to gather the error comparison from 4 literature for resistance and propulsion. ForPeriod A, where the work related to resistance and propulsion are relatively lesser than the other periods considered, maximum error reported was more than 10% in the CFD workshop held at Gothenburg in the year 2000 [84]. After 2003, there was a surge in the published data with improved accuracy of validation results. In the 26th ITTC conference proceedings held in 2011, the deviation of the results with the earlier simulations were much reduced and it is reported that the percentage error for resistance and propulsion is in the range of 3% to 8%. The committee recommended the usage of CFD as an efficient design tool due to the consistency in the results.The accuracy of the results was evolving since 2014 and it has reached in a stage that in the near future the CFD technique may replace the experimental techniques especially for resistance and propulsion.

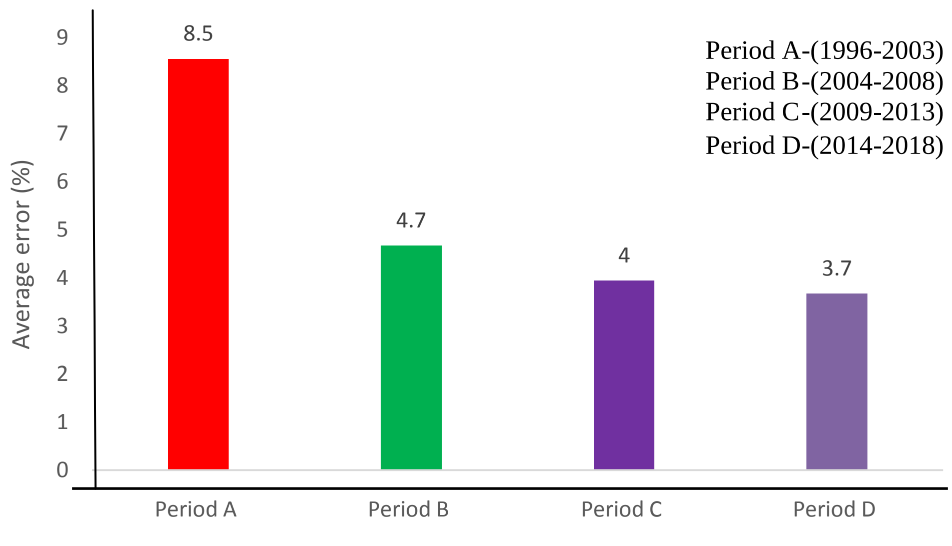

Fig 13: Average error comparison for resistance and propulsion using CFD method

4.1 Seakeeping and Maneuvering

Computations of seakeeping and maneuvering is popular as the complex dynamic simulations using CFD is becoming easier with reasonable results. The dynamic simulations are becoming more popular with complex geometries including merchant ships, surface combatants, high speed crafts since the volume of grid generated varies from tens to hundreds of million. Better and finer results are nowadays possible for seakeeping and maneuvering simulations due to this capability of computing. Last decade, the results obtained for seakeeping tests for head sea conditions for obtaining the 2 DoF motions of heave and pitch and added resistance were unreliable [2] but the progress in physics and dynamic meshing allowed the CFD to provide better results [14]. However, technical improvements are needed to predict the seakeeping response in low speed as well as in random seas. The result obtained for wave and pitch motion with experimental comparison is shown if Fig 14 and 15 respectively.

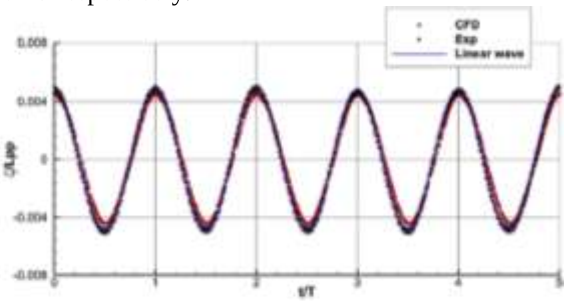

Fig 14: Regular wave generation

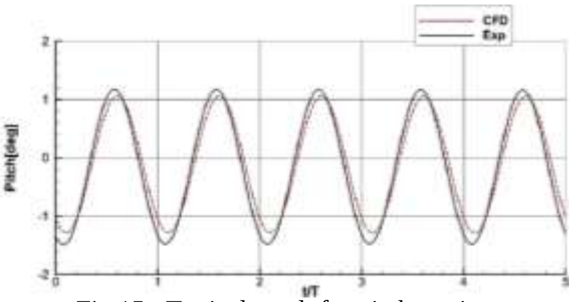

Fig 15: Typical result for pitch motion

The work carried out in seakeeping and maneuvering has been extensively reviewed in the year 2013 and 2014 by Stern [4, 51]. It has been stated that the RANSE based CFD solutions are yet to become mature to provide the results related to waves and highly turbulent scenarios and it is predicted that by 2020 very likely these problems would be tackled for ship hydrodynamics. Recently the research in the field of seakeeping is increasing with better results especially added resistance with varying wave steepness [17, 43], 2 DoF motions of normal hull shapes as well as twin hull and catamarans [62, 85], real simulations of seakeeping studies with rotating propeller using overset grid [36], damaged hull analysis in waves in head sea conditions [86].The maneuvering simulations for estimating the hydrodynamic derivatives in calm water has become popular with good agreement with experimental results, both in shallow and deep-water condition [32]. Recently, the defense research section of Canada has carried out PMM simulations and validated with experiments with approximately 3% error [7].Turning circle simulations with single and twin screw are also nowadays possible with reasonable agreement. However,more studies to be carried out for more accurate results to match the real free running tests with wave interaction [87, 67].A typical test case for turning circle carried out for a twin-screw vessel is shown in Fig 16. Free running simulations for zig-zag maneuver in waves has been reported recently for various wave conditions [89]with reasonable results. Even though many literature discusses the seakeeping and maneuvering studies, in future more validation studies has to be carried out to simulate the real time scenario in waves in order to obtain the effects on the trajectory and course. A comparison on the average error for maneuvering simulations such as straight line test, PMM, turning circle and zig-zag maneuver has been plotted. See Fig 17. It can be inferred from the statistics that the development of maneuvering simulation for last decade till present is developing in gradual pace.A comparison has been made to assess the progress of CFD in ship hydrodynamics in last three decades and tabulated in the Table 5. CFD is continuing with advances in physics, new meshing algorithms and high computing facility. It should be noted that, the progress made in CFD with multiple simulation strategy was not imaginable before ten years has now become routine.Further it is expected that the CFD might advance in coming years and the complexity not feasible today would likely be routine in another ten years.

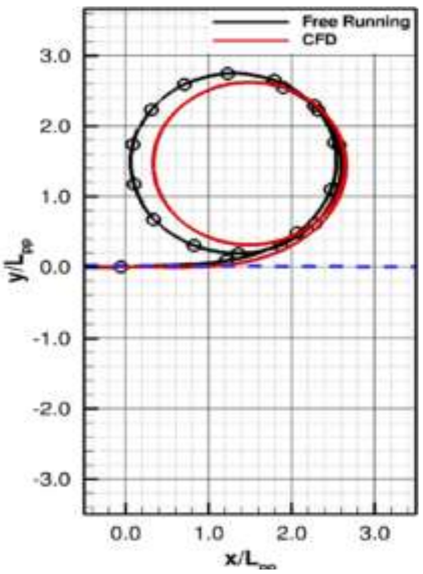

Fig 16: Turning circle simulation and validation with free running test

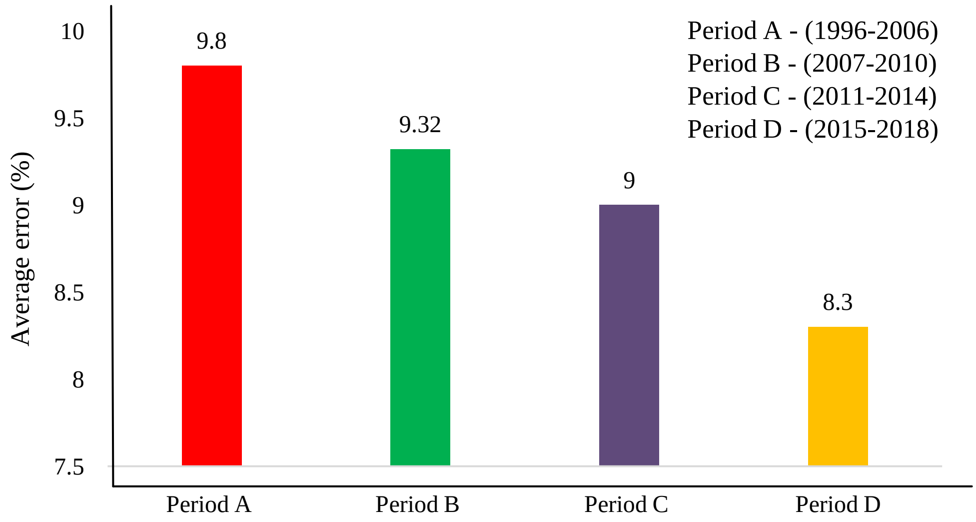

Fig 17: Average error comparison for maneuvering using CFD method (Straight line, PMM, Turning circle, Zig-Zag)

Table 5: Status of CFD in ship hydrodynamics in last three decades

Parameters	Period A (1990-1999)	Period B (2000-2009)	Period C (2010-2018)
Resistance	Assisting existing data	Reasonable validation with experiments	Reliable
Propulsion	Developing stage – able to assist reasonably the existing data	Reasonable validation with experiments	More reliable with overset/dynamic mesh
Maneuvering	Initial stage – few studies reported	Reasonable results for Static maneuver	Reasonable results for Static/dynamic maneuver
Seakeeping	Scarcity in the availability of literature (may be no studies reported)	Reasonable for head sea condition in regular waves	No further major development
Grid number	Million	Tens of millions	Hundreds of millions
Free surface modelling	Surface tracking method	Surface tracking method / Surface capturing method	Mostly surface capturing method

V. CONCLUSION

The present paper reviewed the progress reported in ship hydrodynamics from CFD perspective. Nowadays the application of parallel computation is growing which makes the CFD results looks more promising. It has been observed from the literature that the computations for the resistance and propulsion has become consistent in providing accurate results within the range of 2% to 5%. For maneuvering simulations, reasonable results are achieved for static and dynamic maneuvers but many more aspects of ship steering in waves has to be validated. Simulations using CFD provides encouraging results for motions in head sea conditions in regular waves. But for other heading conditions such as beam sea and following seas, the progress is not evident. The actual condition of motion studies considering the 6-DoF cases are also in evolving stage.

Although CFD has been largely been done for resistance and propulsion studies in ship hydrodynamics, it is nowadays rapidly progressing for very complicated dynamic areas of seakeeping and maneuvering. The progress in CFD simulations has made it possible to provide similar information at par with model tests and moreover it provides the information other than the experiments. There are limitations just like the experiments, but the advance in high performance computing makes the CFD simulations much attractive as seen from time and cost perspective.

REFRENCES

[1]. L Larsson, F Stern, M Visonneau (2013) CFD in Ship Hydrodynamics—Results of the Gothenburg 2010 Workshop

[2]. The Specialist Committee on Computational Fluid Dynamics, Proceedings of 26th International Towing Tank Conference , 2011

[3]. Practical guidelines for ship CFD applications, International Towing Tank Conference (ITTC) recommended procedures and gudelines , 2014

[4]. F Stren, J Yang, Z Wang, H Sadat-Hosseini, M Mousaviraad, S Bhushan, T Xing (2013) Computational ship hydrodynamics: Nowadays and way forward, International Shipbuilding Progress 60 3–105

[5]. J F Thompson, Z.U.A.Warsi, C.W.Matin Numerical grid generation

[6]. B Guo, S Steen, G Deng (2012) Seakeeping prediction of KVLCC2 in head waves with RANS, Applied Ocean Research Vol.35

[7]. C Oldfield, M Larmaei1, A Kendrick, K.Taggart (2015) Prediction of warship manoeuvring coefficients using CFD ,Report by Defence Research and Development Canada

[8]. Y Demirel, O Turan, A Incecik (2017) Predicting the effect of biofouling on ship resistance using CFD, Applied Ocean Research 62

[9]. Handbook of grid generation. Thompson, Soni, Weatherill, CRC Press

[10]. P Kanti1, Shyam chandran C (2016) A Review Paper on Basics of CFD and Its Applications,National Conference on Advances in Mechanical Engineering Science

[11]. A Kolakoti1, T Bhanuprakash , H N Das (2013) ,CFD analysis of controllable pitch propeller used in marine vehicle, G.J.E.D.T, Vol. 2(5):25-33 , September-October

[12]. Seakeeping Committee Report, Proceedings of 24th International Towing Tank Conference (ITTC),Edinburgh, Scotland,2005

[13]. Y K Demirel, M Khorasanchi, O Turan, A Incecik, M P Schultz (2014) A CFD model for the frictional resistance prediction of antifouling coatings, Ocean Engineering 89 21–31

[14]. Y C Kim, K S Kim, J Kim, Y Kim, Il-Ryong Park, Y H Jang (2017) Analysis of added resistance and seakeeping responses in head sea conditions for low-speed full ships using URANS approach, International Journal of Naval Architecture and Ocean Engineering

[15]. S Mousaviraad, S Cook, P Carrica, Y Toda, F Stern (2012) Complimentary EFD and CFD on Effects of Headwinds on Towing Tank Resistance and PMM Tests for ONR Tumblehome , 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden

[16]. The Specialist Committee on CFD in Marine Hydrodynamics , 26th International Towing Tank Conference, 2014

[17]. M Kim,, O Hizir, O Turana, A Incecik (2017) Numerical studies on added resistance and motions of KVLCC2 in head seas for various ship speeds, Ocean Engineering 140 466–476

[18]. H Liu, N Ma, X Gu.(2015) Maneuvering prediction of a VLCC model based on CFD simulation for PMM tests by using a circular water channel.Proceedings of the ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering OMAE2015 May 31-June 5, 2015, St. John's, Newfoundland, Canada

[19]. Guideline on use of RANS tools for manoeuvring prediction. International Towing Tank Conference Recommended Procedures and Guidelines 2011

[20]. Z Shen, D Wan, P M Carrica. (2015) Dynamic overset grids in Open FOAM with application to KCSs elf-propulsion and maneuvering. Ocean Engineering108(2015)287–306

[21]. P Kellett, O Turan, A Incecik (2013) A study of numerical ship under water noise prediction, Ocean Engineering 66 113–120

[22]. Resistance and Flow Committee Final Report and Recommendations of the 21st International Towing Tank Conference 1996

[23]. H Liu, N Ma, X Gu.(2016) Numerical prediction of ship hydrodynamic derivatives in close proximity to a vertical bank and maneuvering stability analysis.Proceedings of the ASME 2016 35th International Conference on Ocean, Offshore and Arctic Engineering OMAE2016 June 19-24, 2016, Busan, South Korea

[24]. T Takai, M Kandasamy , F Stern (2011) Verification and validation study of URANS simulations for an axial waterjet propelled large high-speed ship, Journal of Marine Science and Technology 16(4) , 434–447

[25]. S Kaidi , H Smaoui , P Sergent (2017) Numerical estimation of bank-propeller-hull interaction effect on ship manoeuvring using CFD method , Journal of Hydrodynamics 29(1) 154-167

[26]. H Raven, A Ploeg (2007) Extending the benefit of CFD tools in ship design and performance prediction, 7th International Conference on Hydrodynamics (ICHD)

[27]. T Xing, P Carrica , F Stern (2008) Computational towing tank procedures for single run curves of resistance and propulsion, ASME J. Fluids Eng. 130(10) 1–14

[28]. H Hosseini, F Stern, A Olivieri, E Campana, H Hashimoto, N Umeda, G Bulian, A Francescutto (2010) Head-waves parametric rolling, Ocean Engineering 37(10) 859–878.

[29]. F Ismail, P Carrica, T Xing, F Stern (2010) Evaluation of linear and nonlinear convection schemes on multidimensional non-orthogonal grids with applications to KVLCC2 tanker, International Journal Numerical Methods in Fluids 64 850–886

[30]. K Mizzi, P Kellett, Y Demirel, R Martin, O Turan (2015) HPC and CFD in the marine industry: past, present and future,15th Proceedings of The 3rd International Conference On Exascale Applications And Software, Edinburgh

[31]. R R Shenoi, P Krishnankutty, R P Selvan. (2016) Study of manoeuvrability of container ship by static and dynamic simulations using a RANSE-based solver.Ships and Offshore Structures, 2016 Vol. 11, No. 3, 316–334

[32]. S He, P Kellett, Z Yuan, A Incecik, O Turan, E Boulogoris (2016) Manoeuvring prediction based on CFD generated derivatives, Journal of Hydrodynamics 28(2) 284-292

[33]. Practical Guidelines for Ship Self-Pro-pulsion CFD, ITTC – Recommended Procedures and Guidelines, International Towing Tank Conference (ITTC) 2014

[34]. A Kolakoti,T V K Bhanuprakash, H N Das.(2013) CFD analysis of controllable pitch propeller used in marine vehicle

[35]. Pratical Guideline for ship Resistance CFD, ITTC- Recommended Procedures and Guidelines, International Towing Tank Conference (ITTC) 2014

[36]. Z Shen, P Carrica, D Wan (2014) Ship motions of KCS in head waves with rotating propeller using overset grid method,Proceedings of the ASME, 33rd International Conference on Ocean, Offshore and Arctic Engineering (OMAE) June 8-13, California, USA

[37]. G Dhinesh, K Murali, V A Subramanian. (2010) Estimation of hull-propeller interaction of a self-propelling model hull using a RANSE solver.Ships and Offshore Structures Vol. 5, No. 2, 2010, 125–139.

[38]. J Choi, K Min, J H Kim, S B Lee, H W Seo(2010) Resistance and propulsion characteristics of various commercial ships based on CFD results, Ocean Engineering 37 549-566

[39]. H Kim,C Yang(2010), A new surface modification approach for CFD-based hull form optimization, Journal of Hydrodynamics 22(5) 520-525

[40]. General uncertainty analysis in CFD verification and validation methodology and procedures. Recommended Procedures and Guidelines, International Towing Tank Conference (ITTC) 2008

[41]. General uncertainty analysis in CFD, Guidelines for RANS codes, Recommended Procedures and Guidelines, International Towing Tank Conference(ITTC) 1999

[42]. E Campana , D Peri, Y Tahara, F Stern (2006) Shape optimization in ship hydrodynamics using computational fluid dynamics, Computer Methods in Applied Mechanics and Engineering 196(1) 634-651

[43]. A Fitriadhy, N S Razali ,N AqilahMansor (2017),Seakeeping performance of a rounded hull catamaran in waves using CFD approach,Journal of Mechanical Engineering and Sciences

[44]. D Oliveira, A I.Larsson, L Granhag (2018) Effect of ship hull form on the resistance penalty from biofouling, The Journal of Bioadhesion and Biofilm Research

[45]. Y Jin, S Chai, J Duffy, C Chin, N Bose, L Sun (2016) URANS prediction of ship hydrodynamics in head sea waves at zero forward speed with model testing validation, Proceedings of the ASME 35th International Conference on Ocean, Offshore and Arctic Engineering (OMAE) June 19-24, Busan, South Korea

[46]. R Muscari, G Dubbioso, M Viviani, A Mascio.(2017) Analysis of the asymmetric behaviour of propeller–rudder system of twin screw ships by CFD.

[47]. A Koop, H Cozijn, P Schrijvers, G Vaz (2017) Determining thruster-hull interaction for a drill ship using CFD, Proceedings of the ASME 36th International Conference on Ocean, Offshore and Arctic Engineering (OMAE) June 25-30, Trondheim, Norway

[48]. C T Lin, T Y Lin, C W Lin, Y W Hsieh, L Lu, C Y Hsin (2017) Investigation of the seakeeping performance of twin hull vessels bydifferent computational methods,10th International Workshop on Ship and Marine HydrodynamicsKeelung, Taiwan, November 5-8

[49]. T Xing, P Carrica , F Stern (2011) Developing streamlined version of CFDShip-Iowa-4.5, IIHR Technical Report No. 479

[50]. M Mousaviraad, S Bhushan , F Stern (2012) CFD prediction of free-running SES/ACV deep and shallow water maneuvering in calm water and waves in Proceeding of MARSIM

[51]. F Stern , Z Wang, H Hosseini, M Mousaviradd, S Bhushan, M Diez , S H Yoon, P C Wu, S M Yeon, T ogan, D H Kim, S Volpi, M Conger, T Michael, T Xing, R Thondal, J Grenestedt (2014) Recent progress in cfd for naval architecture and ocean engineering .Proceedings of the 11th International Conference on Hydrodynamics (ICHD) October 19 – 24, Singapore

[52]. P Carrica, R Wilson, R Noack, F Stern (2007) Ship motions using single-phase level set with dynamic overset grids, Computers & Fluids 36 1415–1433

[53]. A Ploeg ,H Raven (2007) Accuracy of viscous resistance computations for ships including free surface effects, International Conference on Computational Methods in Marine Engineering (MARINE)

[54]. Z Zlatev and E Milanov, V Chotukova, N Sakamoto , F Stern (2009) Combined model-scale EFD-CFD investigation of the maneuvering characteristics of a high speed catamaran, 10th International Conference on Fast Sea Transportation (FAST), Athens, Greece

[55]. S Bhushan, T Xing, P Carrica , F Stern (2009) Model and full-scale URANS simulations of Athena resistance, powering, seakeeping, and maneuvering, Journal of Ship Research53(4) 179–198

[56]. D Rijpkema, B Starke, J Bosschers (2013) Numerical simulation of propeller-hull interaction and determination of the effective wake field using a hybrid RANS-BEM approach, Third International Symposium on Marine Propulsors smp, Tasmania, Australia, May

[57]. Zhang Z.-R (2010), Verification and validation for RANS simulation of KCS container ship without/with propeller, Journal of Hydrodynamics

[58]. Hino, T. Ohashi, K ,Kobayashi, H (2010) Flow simulations using Navier-Stokes solver surf, Proceedings of Gothenberg Workshop.

[59]. C Crane (1987) Maneuvering trials of 278000-DWT tanker in shallow and deep waters, Trans. SNAME87, 251-28

[60]. F Stern, K Agdrup (2008) Workshop on verification and validation of ship maneuvering simulation methods ,SIMMAN , Copenhagen, Denmark

[61]. A Aksenov, S Zhluktov, V Llyin, D Silaev, S Kharchenko, A Pechenyukm (2017) Investigating the Problems of Ship Propulsion on a Supercomputer, Ivannikov ISPRAS Open Conference (ISPRAS)

[62]. Chun-Ta Lin, Tsung-Yueh Lin, Cheng-Wen Lin, , Yu-Wen Hsieh, Ling Lu ,Ching-Yeh Hsin.(2017) Investigation of the Seakeeping Performance of Twin Hull Vessels by Different Computational Methods

[63]. .Z Shen, D Wan, P M Carrica (2015) Dynamic overset grids in Open FOAM with application to KCS self-propulsion and maneuvering, Ocean Engineering108 287–306

[64]. R Wilson, F Stern, H Coleman, E Paterson(2001) Comprehensive approach to verification and validation of CFD simulations-Part 2: Application for RANS simulation of a Cargo/Container Ship, Journal of Fluid Engineering 123(4) 803-810

[65]. P Carrica, H Hosseini, F stern (2012) CFD analysis of broaching for a model surface combatant with explicit simulation of moving rudders and rotating propellers, Computers and Fluids 53 117–132.

[66]. T Castiglione, F Stern, S Bova , M Kandasamy (2002) Numerical investigation of the seakeeping behavior of a catamaran advancing in regular head waves, Ocean Engineering 38(16) 1806–1822

[67]. G Dubbioso, D Durante, A Mascio, R Broglia (2016) Turning ability analysis of a fully appended twin screw vessel by CFD Part II : Single vs .twin rudder configuration

[68]. A Masico, G Dubbioso, R Muscari, M Felli (2015) CFD Analysis of Propeller-Rudder Interaction, Proceedings of the 25th International Ocean and Polar Engineering Conference Kona, Big Island, Hawaii, USA, June 21-26

[69]. F Stern , R Wilson, H Coleman, E Paterson(2001) Comprehensive approach to verification and validation of CFD simulations-Part 1: Methodology and Procedures. Journal of Fluids Engineering 123(4) 793-802

[70]. Recommended Procedures and Guidelines, Practical Guidelines for Ship Resistance CFD ,International Towing Tank Conference (ITTC) 2011

[71]. M Shahid ,D Haung(2011) Resistance calculation of trimaran hull form using computational fluid dynamics, Fourth International Joint Conference on Computational Sciences and Optimization

[72]. Y Tahara, Satoshi T, Tokihiro K(2006) CFD-based multi-objective method for ship design, International Journal Numerical Methods in Fluids 52(5) 499-527

[73]. M Salas , R Luco, N Browne, M Lopez(2014) Experimental and CFD Resistance Calculation of a Small Fast Catamaran

[74]. S Sherbaz, W Duan (2014) Ship trim optimization: assessment of influence of trim on resistance of MOERI container ship, Hindawi Publishing Corporation. Scientific World Journal Volume

[75]. A Fitriadhy , S Azmi , N Mansor , N Aldina.N (2017) Computational fluid dynamics investigation on total resistance coefficient of a high-speed "deep-V" catamaran in shallow water, International Journal of Automotive and Mechanical Engineering Vol 14, 4369-4382 June

[76]. H Hosseini , P C Wu, P Carrica, H Kim, Y Toda, F Stren (2013) CFD verification and validation of added resistance and motions of KVLCC2 with fixed and free surge in short and long head waves, Ocean Engineering 59 (2013) 240–273

[77]. A Alho (2012) Validation of CFD predictions of the hull resistance and the wave system of a catamaran, Proceedings of the ASME ,31st International Conference on Ocean, Offshore and Arctic Engineering (OMAE), Rio de Janeiro, Brazil, July 1-6

[78]. Y C Wang (2015) Numerical prediction of resistance of planning vessel with RANS method, 5th International Conference on Computer Sciences and Automation Engineering (ICCSAE)

[79]. I Vinola , R Flay , R Ponzini (2010) Practical CFD to compute resistance, sink and , trim of fully appended sailing yachts, 17th Australian Fluid Mechanics Conference, Auckland, New Zealand, 5-9 December

[80]. S. Bhushan, M Mousaviraad, F Stern , L Doctors (2011) T-Craft resistance and motions, seakeeping in regular waves, 11th International Conference on Fast Sea Transportation (FAST) , Honolulu, Hawaii, USA

[81]. D Sridhar, T Bhanuprakash , H Das (2010) Frictional Resistance Calculations on a Ship using CFD, International Journal of Computer Applications(0975 – 8887) Vol.11

[82]. K Sugalski (2014) Fishing vessel hull design and towing resistance calculation by the CFD methods, Scientific Journals Maritime University of Szczecin 40(112) 27–30

[83]. S Bhushan,T Xing, F Stern (2012) Vortical Structures and Instability Analysis for Athena Wetted Transom Flow with Full-Scale Validation. Journal of Fluids Engineering 134 031201-1-18

[84]. L Larsson, F Stern, V Bertram (2003) Benchmarking of Computational Fluid Dynamics for Ship Flows: The Gothenburg 2000 Workshop. Journal of Ship Research, Vol. 47, No. 1, March

[85]. B Aktas, F Prini, S Benson (2017) Full-scale unsteady RANSE CFD seakeeping simulations of a high-speed craft,20th Numerical Towing Tank Symposium. 2017, Wageningen, Netherlands

[86]. J Seo ,S Rhee (2016) Experiments and CFD analysis on safe-return-to-port of a damaged ship in head and following seas,Proceedings of the ASME Fluids Engineering Division Summer Meeting (FEDSM)

[87]. R Broglia , G Dubbioso , D Durante , A DiMascio (2015) Turning ability analysis of a fully appended twin screw vessel by CFD PartI:Single rudder configuration

[88]. M Kandasamy, S Ooi, P Carrica, F Stern (2010) Integral force/moment water-jet model for CFD simulations, Journal of Fluids Engineering132 1-9

[89]. J Wang, L Zou ,D Wan (2018) Numerical simulations of zig zag maneuvering of free running ship in waves by RANS –overset grid method ,Ocean Engineering.

Abhiroop K" A review on the advancement of CFD technique in ship hydrodynamics" International Journal of Computational Engineering Research (IJCER), vol. 08, no. 12, 2018, pp 38-51