State of the Art Review of the Application of Computational Fluid Dynamics for High Speed Craft

H. A. Ahmad,a,∗ and A. F. Ayobb

aSchool of Ocean Engineering, Universiti Malaysia Terengganu, Terengganu, Malaysia
*Corresponding author: ahmad.faisal@umt.edu.my

ABSTRACT

Computational Fluid Dynamics (CFD) field of research continues to advance with several new accomplishments, especially in the field of high speed craft design. This paper presents the application of advanced CFD simulations reported in the literatures in terms of software tools available, hull form optimization, resistance, and seakeeping analysis and propulsion system. It aims to review case studies of several methods utilized to conduct operational simulations that result in useful performance prediction of high speed craft. The simulations reviewed in this paper are referring to the research reported in the literatures over the last decade on patrol boats, catamarans, rescue boats and so on. Some notable analysis were discussed, e.g. resistance and seakeeping analysis, propulsion system, hull form optimization and multi physics. We conclude that based on the vast resources available in the literature, CFD is one the best tool for designing high speed craft, however such work could be impacted more by its combination with optimization methods.

KEYWORDS: Computational Fluid Dynamics; High Speed Craft; Simulation; Resistance and Seakeeping Analysis

1.0 INTRODUCTION

Computational Fluid Dynamics or CFD is a branch of fluid mechanics that involves numerical analytic process and algorithms to simulate fluid flow around an object. These state-of-the-art methods had a vast means of application, due to the availability of numerical codes and the capability of the computational performance, with the possibility to generalize certain algorithms to more intricate physical quandaries [1]. Additionally, the naval architecture society has initiated to extensively apply these methods for predicting stable and unstable performance of marine vehicles. Due to time and resources constraint in industry, some CFD tools in hydrodynamic optimization were suggested in the literature, essentially to decrease wave patterns and calm-water resistance [2], such as in the works reported in [3-5].

Optimization is a field of research that concern about the determination of the best design with respect to the desired design characteristics. Typically, optimization of a ship hull starts by selecting the best appropriate objective function, determination of the most suitable numerical modelling of hull types (displacement craft or high speed planning craft), selection of an effective numerical tool, and execution of optimization process for one or more objective problems [6]. Although the experimental approach is still utilizable, it has its own restrictions e.g. cost, space, and man-hours [7, 8]. The work that utilize CFD for optimizing high speed craft in preliminary design stage has been reported in [9], however such application is still limited in the literature.

High speed crafts (HSC) [10] are high-speed marine vessels with normally range from small leisure boats to huge combatant crafts [11, 12]. A lot of research done on optimizing high speed craft is to minimize the resistance occurred and to maximize the speed, however most of the performance prediction equations are limited to empirical equations [135-137]. Optimization algorithms are widely used to optimize high speed craft hull form [10, 13-18]. Sekulski et al. [19, 20] applied Genetic Algorithm (GA) to analyze structural weight and painted surface area on a high-speed catamaran with factors, such as plate size and density, and also spaces between stiffeners. Speed maximizing researches [5, 21-27] do act as an important paradigm in high speed craft optimization.

2.0 SHIP DESIGN SPIRAL

The early form of a ship commonly created through four phases:
concept; preliminary; contract; and detail design. The design spiral (Figure 1) picturize the early design processes in which design events are sequentially addressed in order to achieve a converged, feasible ship design. At each section, a certain number of data is added, details are illustrated, and engineering computations are worked out. Work proceeds around the design spiral until the design is finalized.

![Figure 1: Ship Design Spiral [28]](Image)

Evans et. al. [29] made a major contribution of explaining the ship design spiral process through visualization and modelling. As the research progressed, a lot of researchers come up with new ideas and notions on limitations of the process such as life-cycle cost, modularity and integrated operation and ways to overcome them [30]. Mizine et. al. [13, 17, 31, 32] introduced a method called Multi-Level Hierarchical System (MLHS) by using high-fidelity design and analysis tools during early stage design. A design framework is proposed by Jang et. al. [33] via BRIX workflow manager system which mirrored a scenario based design prone to human glitch and accidental setback under complicated interactions among several design teams. Hefazi et. al. applied multi-disciplinary design and optimization (MDO) approach that determine propulsion, cost, structural loads and seakeeping, which then is optimized using multi-objective optimization method such as Multi-Objective Genetic Algorithm (MOGA) [10, 14-16, 34]. Classical ship design spiral has many steps that may lead to time-consuming for designers to get the best design. An integrated system such as Computer Aided Engineering (CAE) is proven to minimize the analysis time and also reduces the complexity associated with CFD analysis [35-38]. Nadia et al. [39-44] works onto developing systems architecture using Model-Based Engineering System (MBSE) that provides numerous changes, decisions, and results.

### 3.0 SOFTWARE TOOLS AVAILABLE

Computational Fluid Dynamics (CFD) is a method to apply partial differential equations system by a set of algebraic equations, which can be solved within powerful mainframe computer. Some of the well-known numerical tools used widely in the maritime research community are ANSYS CFX, Maxsurf, CFDSHIP-Iowa, Shipflow and many more [2]. Lotfi et al. [45] employed the ANSYS CFX numerical tool to determine the aspects of a one transverse step planing hull.

An empirical method presented by Svahn et al. [46] was used to determine the initial draft and trim angle, then the algorithmic correction was added for the end outcome. The steady-state simulation on comparing numerical performances between stepped and non-stepped hull of a planing hull has been carried out using CFX software [12, 47] as displayed in Figure 1. Shipflow has been generally used for ship resistance calculations, both potential and viscous flows [10, 18, 48-51]. The potential flow calculations module within this numerical tool is based on a 3D Rankine sources distribution method, with the sources being distributed over the vessels over the free and wetted surface. To increase the calculation’s certainty, an iterative nonlinear solution scheme is used taking into account the effect of running trim and sinking, for greater speed. This numerical tool used by Moraes et al. [52] to calculate wave resistance on high speed catamarans by varying hull distance and water depth.

FlowVision is a modern CFD tool occupied with the Finite Volume (FV) method. Aksenov et al. [1] mentioned that this state-of-the-art tool serves all the crucial skills and technologies to be fully utilized in ship design work. A team of Scottish engineers also had used FlowVision to analyze the aero and hydrodynamics of sailing boat using a different length of wing-sail cruising at more than 60 knots [53]. Vishnevsky et al. [54] investigate the hydrodynamics behavior and propulsion of high speed vessel by alaying applications of fixed pitch propeller and variable pitch propeller. The numerical approaches derived from Reynolds Averaged Navier-Stokes (RANS) equations, are among of the most dependable approaches to determine a hull’s motion. University of Iowa’s IIHR – Hydrosience & Engineering has developed a general purpose CFD simulation software namely as CFDSHIP-Iowa to support research done in universities and industries. Modern analysis viewed that the moment and resistance due to air inside the cavity is compelling for seakeeping scenario but not calculated in the early numerical investigations. [6].

The method used in COMET is of finite-volume-type and applied control volumes (CVs) with an irregular number of uncomplied meshes [55-58]. The method is parallelized by
domain decomposition in both space and time and is thus well suited for 3-D flow computation with free surfaces [59, 60]. To reduce the motions predicting process, Azcuen et al. [59] combined the fluid flow with the body motions by spanning the Navier–Stokes solver COMET with a body motion module.

Figure 2: Seakeeping simulation [59]

Subramanian et al. [11] established two tunnels to a planing hull with the aim of choosing suitable propeller and decreasing the shaft angle. Likewise, the research involves analyzing the effects on the drag and lift forces. The FLUENT tool is used to carry out the FVM for the RANS equations [61-63]. The outcome are then compared with Savitsky’s equation. Yousei et al. [64] utilized the FVM based FLUENT software to examine the flow around a planing monohull and a two tunnels hull. Ghassabzadeh and Ghassemi et al. [65] applied the numerical tool to simulate a multi-hull tunnel vessel operating in calm water.

MAESTRO is a design building tool for constructing any type of 3-dimensional model. The flow solver MAESTRO-Wave is used to determine motions and wave impacts by applying strip theory and panel approach. A seakeeping analysis done by the Australia’s Department of Defence [4] by utilizing basic developed numerical implement PANSHP. Result gained is then compared with the infamous state-of-the-art software Maxsurf which resulted that Maxsurf resistance methodologies are more precise than PANSHP.

PANSHP where else has the capability to simulate the usage of trim tabs to the total resistance than Maxsurf. The PANSHP numerical tool is proved to be the most precise when comparing the full-scale sea tribulation to the numerical result [7, 66-68]. Sayeed et al. investigate on applying precise numerical models of planing hull motion in waves by generating an actual environment for training Fast Rescue Craft (FRC) operators in an authentic-time simulator. A computer program Planing Hull Motion Program (PHMP) [69, 70] has been invented focusing on the 2D non-linear time domain divests theory. The numerical models were tested in an FRC simulator provided by Virtual Marine Technology Inc. and approved to be compiled [71].

4.0 RESISTANCE & SEAKEEPING ANALYSIS

4.1 Resistance Prediction

Forces acting upon a planing craft is drag numbers poised on the wetted surface, the buoyant force, transom pressure and air resistance, the weight of the vessel and the propulsor’s thrust [72]. Hydrodynamic forces are considered one of the most influential parameter for resistance prediction and minimization of a planing high speed craft [26, 73-80] that may leads to changes of draught and displacement of the hull. The prediction of resistance of planing craft has been the objective of many investigations. Özüm et al [73] applied the 6-DOF Motions to retrieve the vertical plane motions and lift force of a high speed craft, simulated in STARCCM+ and comparing the results with the Savitsky resistance prediction statistical results. Dynamic mesh method is also has been proven to be useful to forecast the resistance of a high speed marine vehicle by focusing on the hull gesture [9]. High speed river craft, namely ferry, is analyzed using the dynamic mesh concentrating on heave and pitch motions, and total resistance number resulted with a Froude number between 0 to 4 [81] as shown in Figure 3.

Figure 3: Pressure contour on the bottom hull at Froude number = 2.76 [82]

High speed craft also has been designed with tunnels at the bottom of the hull and being analyzed whether or not it can reduce the drag force acted upon the bottom hull [82-86]. With the present tunnel, forward flow is minimized to allow the vessel experiencing optimum trim condition [11]. With two tunnels designed at the hull’s underwater area, resulted in drag force reduction of the altered hull [64]. Subramanian et al. [11] investigated pressure and resistance acted upon one chine planing hull by using two hull forms, one with a tunnel (also called “propeller pockets”) and one without, and finally compared the results with experimental test results.

Brizzolara et al. [87] used the CFD codes to predict planing surfaces by applying a wedge shaped planing hull to CFD analyses and varying the running trim angle systematically and compared the results with model tests and semi-empirical theories. Nowadays, the high speed craft’s market demands catamarans of different dimensions, designed to be high speed with low resistance. Thus, it is fundamental for hull resistance optimization in designing a marketable high speed catamaran hull [88, 89].

Drag-Lift ratio of planing hulls increases with the increment of speed [75, 90]. An empirical approach recently has been applied to determine the behaviour of stepped hulls. Sahn et al. [46] combined these equations with Savitsky’s method for current planing hulls and developed a semi-empirical method to determinet the stepped planing hull behavour.

4.2 Seakeeping Analysis

Recently, industrial ship design activities opted to include the assessment of seakeeping behavior aspect [2]. Latest computational method enabled the designer to analyze the seakeeping behavior of ship in preliminary stage [91]. Seakeeping analysis can be divided into several types which are investigation the ship motions and workability limits, parametric rolling and non-linear roll effects, hydrodynamic interactions with environmental forces such as wind and waves, and also anti-rolling tank evaluation[92] (mostly for offshore service vessels) [93]. Seakeeping and calm water measurements tests have been
carried out for a typical high speed catamaran; data collected is a valuable set for both hydrodynamic analysis and CFD validation [94]. CFD analysis practices and techniques have been developed for seakeeping analysis. CFD technology has become modernized and dependable numerical methods for high speed craft seakeeping analysis [5, 21, 24, 25, 95-101]. Figure 4 shows the result gained by Prini et al. by performing an analysis on a numerical model of a lifeboat to calculate the pressure resulted, body motions and the impacts in typical waves which resulted that the highest heave and pitch feedback at wavelengths that are greater than the vessel length [102].

A couple of research also have been done on simulating high speed craft in a regular wave using the Reynolds Averaged Navier Stokes Equations Volume of Fluid (RANSE VOF) solver [22, 23, 103, 104]. International Maritime Organization (IMO) puts a concern on the safety of fishing vessel due to the fact of at least 24,000 human losses have been reported annually [105]. Some researchers focusing on to optimize the seakeeping performance of the fishing vessel based on to minimize human loss at sea cases [3, 14]. Tello et al. [106] numerically investigated the seakeeping performance of a fishing vessel by considering it operating in sea state 5 and 6. Motions such as roll, pitch and heave will affect the stability of the fishing vessel, thus some research is done whether to optimize the hull form or moving the metacentric height for example, in order to stabilize the boat while operating in certain sea condition [107, 108]. Experimental studies are also another options on maximizing the seakeeping value of a craft due to environmental factors such as injecting air from aft surface to the step by Lay et al. [109] and installing flap at a yacht hull by Day et al. [110].

4.3 Propulsion Design & Analysis

Most high speed craft are designed to be installed with propulsion systems such as outboard engine, inboard engine, water jets and surface drive. Outboard engine as in Figure 5(A) is like marine automobile engine installed at the boat’s transom [111]. It does require less maintenance, no winterizing, and easier to work with the outboard outside the boat. Outboard engines also can behave like rudders when operating at an angle of attack and provide turning forces to the thrust [112]. An inboard motor/engine is the diesel powered propulsion system installed within the boat hull, usually connected to a propulsion screw by a driveshaft [113] as in Figure 5(B). Waterjets are also chosen to be an ideal propulsion to be used for vessels operating in waters with restricted depths as they present a solution with minimum draught and protected propulsion as in Figure 5(C). Water jets are very commonly used for speeds between 30 and 45 knots but they sure can propel a vessel even beyond 60 knots. Many catamaran ferries rely on water jets [114]. For surface drive, it is actually about efficiency. As the propeller break the water’s surface, giving fewer loads run by the engine as shown in Figure 5(D). With half of the propeller rotates above water surface, it propels the boat faster than one with submerged props [115].

The electric propulsion system is now practically being used in high speed craft industry. For outboards, they were built using state-of-the-art components to minimize losses when interactions between components occur e.g. between motor and propeller. This electric propulsion causes no water pollution with exhaust gases, oil or petrol [120]. Choi et al. [121] present the usage of a polymer electrolyte membrane (PEM) fuel-cell-battery hybrid system for the propulsion of a high speed ferry. An experimental study is conducted on a ferry using hybrid fuel cell propulsion system and is compared to a three DOF total ship system simulator developed in MATLAB/Simulink environment [122].

Propeller is also among of the important components in achieving maximum speed for high speed craft. Submerged fixed pitch propeller (FPP) are mostly used for small and medium vessels. For controllable pitch propeller (CPP), it can provide best maneuverability for vessels up to 35 knots [114] although the installation work is more expensive and has higher maintenance...
requirements compared to FPP. Surface piercing propeller are a special type of FPP. Small interceptor vessels owned by enforcement agencies around the world are well-known to use surface piercing props since the speeds achieved can go up to 80 knots [114]. Zimmerman et al. [123] investigate induced velocity field for a propeller operating in a homogenous inflow field using ANSYS Fluent. The results gained show that the modelling approach enables simple employment of varied propeller types and improves computational capacity with required time amount compared to the complex propeller model. Afshar et al. [124] investigate the aerodynamics tampering of a propeller to the tail of a flying boat using CFD. The idea to minimize hydrodynamic forces acting to a monohull has also been approved by the usage of waterjet propeller with rudder [112]. A research also has been conducted by generating numerical simulation of a marine propeller using ANSYS CFX to figure out the open water performance of this propeller for different operating conditions [125]. As known, any peculiar act of an object can be altered due to vibration when encounters fluid. A method called First Order and Second Reliability Methods (FORM and SORM) is numerically studied and is then being applied on an immersed boat propeller to solve the vibrating issue [126]. A study done by Egerton et al. [127] shows the virtual experiment of the propeller velocity and early plume spread formed by propellers in the software code FLUENT. Chapple et al. [128] discussed on the contribution of CFD towards design of ringed propellers for outboard motor application, refer Figure 6.

Figure 6: Ringed propeller[128]

4.4 Hull Form Optimization

Ship hull optimization is classified as a multi variable and objective issue with nonlinear constraints [129]. Grigoropoulos et al. [130] respectively performed planing hull form optimization using wash waves and selected dynamic responses. Biliotti et al. [131] attempted to improve a patrol craft design based on empirical method containing different wave profile in conjunction with the water line, wave resistance, displacement and broad seakeeping operability indicator. Campana et al. [132] meanwhile used reduction of wave resistance as an objective and using a constant upper limit as an inequality boundary for the motion counteraction. Recently, Bagheri et al. [133] showed optimization work done based on acceleration at the bow of the vessel in regular head waves, while Kostas et al. [133, 134] used a T-spline based geometry for resistance optimization.

Several algorithms are derived to be used in ship hull optimizing work such as Non-Dominated Sorting Genetic Algorithm-II (NSGA-II) and Infeasibility Driven Evolutionary Algorithm (IDEA) which are integrated in the planing craft optimization framework [135-137]. Recent study by Khosravi et al. [138] showed the implementation of genetics algorithm (GA) to optimize the seakeeping value of the vessel body line. Jeong et al. [139] presented a framework using a multi-objective genetic algorithm (MOGA) and applying it to hull form optimization by exploring the minimum wave drag configuration under a certain speeds. As viewed in Figure 7, optimum hull geometry is produced by applying optimization algorithm Simultaneous Hybrid Exploration that is Robust, Progressive and Adaptive (SHERPA) within the numerical tool Optimate+ [140].

Zhang et al. [141, 142] optimized the hull form by utilizing the Rankine source method to obtain the wave resistance as the objective function fixating on the optimization of the bow shape. An optimization approach known as SQP (Sequential Quadratic Programming) combined with CFD technique is used to calculate the reduced drag force of a ship hull [143]. Distinctive optimization algorithms can be used to decrease or increase objective functions in the CFD-based hull form optimization. The objective functions are described in terms of the total resistance and seakeeping level. They are graded by the transparent CFD tool and Bales' seakeeping grading approach depicted before [144]. An extensive analysis on seakeeping behavior and hull shape optimization investigated by Cepowski et al. [145, 146] in Poland. He presented a method that makes it viable to determine optimum hull shape of a ferry with regard to certain seakeeping aspects and extra resistance in waves.

4.5 Multi-physics

High speed craft often repeatedly involves in slamming events [147-149] while operating in rough and choppy water environment. Factors like wave topography and ship’s height from the water surface are normally making wave load very difficult to be predicted. High speed craft designer mostly ends up with a timid design by basing only on semi-empirical data. A better understanding of this multi-physics event or complex fluid-structure interactive problem can lead to a fully optimized design [150]. Volpi et al. [151] are currently working on a fully coupled CFD/Finite Element(Fe) Fluid-Structure Interaction(FSI) solver by using ANSYS and inserting the CFD results in as loading for the structural solver by one-way coupling. FSI also can be viewed as hydro elasticity effect on the bottom hull. Figure 8 shows the pressure effects on a numerical model structure with different time-lapse in a slamming impact event [152].

Figure 7: Best design for the basic optimization (in red the new profiles, in blue the old profiles)[140]
The discussions on CFD are concluded with the review of multi-physics simulation and analysis that brings out the prediction analysis on the effect of slamming impact towards high speed craft hull structure.

REFERENCES

Connection To The Hub,” presented at the
Propulsion By Using Of Propellers With Shifted Blade
L. I. Vishnevsky, “Improvement Of High Speed Craft speed-record/
2013.


Scottish sailing engineers have designs on world speed record. Available: https://www.theengineer.co.uk/issues/23-january-2012/scottish-sailing-engineers-have-designs-on-world-speed-record/


100. A. R. Kohansal, H. Ghassemi, and M. Ghaisi, "Hydrodynamic characteristics of high speed planing hulls, including trim effects," *Turkish Journal of Engineering & Environmental Sciences*, vol. 34, pp. 155-170, 2010.


