MINIMISING NUMERICAL VENTILATION IN CFD SIMULATIONS OF HIGH-SPEED PLANING HULLS

Angus Gray-Stephens
Department of Naval
Architecture, Ocean & Marine
Engineering
The University of Strathclyde
Glasgow, Scotland, UK

Tahsin Tezdogan
Department of Naval
Architecture, Ocean & Marine
Engineering
The University of Strathclyde
Glasgow, Scotland, UK

Sandy Day
Department of Naval
Architecture, Ocean & Marine
Engineering
The University of Strathclyde
Glasgow, Scotland, UK

ABSTRACT

Numerical Ventilation (NV) is a well-known problem that occurs when the Volume of Fluid method is used to model vessels with a bow that creates an acute entrance angle with the free surface, as is typical for both planing hulls and yachts. Numerical Ventilation may be considered one of the main sources of error in numerical simulations of planing hulls and as such warrants an in-depth analysis. This paper sets out to bring together the available work, as well as performing its own investigation into the problem to develop a better understanding of Numerical Ventilation and present alternate solutions. Additionally, the success and impact of different approaches is presented in an attempt to help other researchers avoid and correct for Numerical Ventilation.

Interface smearing caused by the simulation being unable to track the free surface is identified as the main source of Numerical Ventilation. This originates from the interface between the volume mesh and the prism layer mesh. This study investigates this interface, presenting a novel solution to prism layer meshing that was found to minimise Numerical Ventilation. Through the implementation of a modified High Resolution Interface Capture (HRIC) scheme and the correct mesh refinements, it is possible to minimise the impact of Numerical Ventilation to a level that will not affect the results of a simulation and is acceptable for engineering applications.

NOMENCLATURE

Fr = Froude Number
 L = Length Between Perpendiculars
 R_n = Reynolds Number
 R_T = Total Resistance
 R_V = Viscous Resistance
 U = Ship Speed
 x = Plate Length
 δ = Boundary Layer Thickness
 Δt = Time Step

INTRODUCTION

The use of Computational Fluid Dynamics (CFD) as a tool for the hydrodynamic assessment of ships has grown considerably in the past 20 years. This is accountable to an increase in the availability of High-Performance Computers, leading to the development of more accurate CFD codes. Additionally, users have become more confident employing CFD as it has become more reliable and established as a design tool. These factors have led to a significant increase in the associated accuracy of simulations, with statistical analysis of the 2010 Gothenburg workshop, which investigated the KCS displacement ship, revealing that all simulations larger than 3M cells were within 4% of the measured resistance data, with a mean comparison error of -0.1%, and a mean standard deviation of 2.1% [1]. With such high confidence levels in the results and its far superior post-processing abilities it is undeniable that CFD is becoming an ever more important tool in the design process of conventional ships.

Unfortunately, the same cannot be said for high-speed craft or other non-conventional vessels for which it is well known that resistance prediction simulations are less accurate. The ITTC noted that it is more difficult to assess the accuracy of CFD for these vessels due to the scarcity of relevant publications [2]. Despite this, they found that for $0.3 < Fr < 0.5$ a mean prediction error of 10% is achievable. This is in line with a number of other papers published, who reported similar levels of error:

- [3] concluded that the level of accuracy for CFD predictions is expected to be around 10%
- [4] do not present their maximum and minimum errors but state that the average error was 10%
- [5] found resistance errors of 4.5 – 9.5%.
- [6] found error in the resistance predictions of 1.9 – 16.7%
- [7] were able to achieve resistance predictions with errors below 10%
- [8] found an error of between 1.2 – 9.3 %. They concluded that they were able to reach comparison error values of below 7.5%

The difficulties in accurately simulating high-speed craft in the fully planing condition are attributable to a number of causes. Brizzolara & Serra reason that these difficulties in resistance prediction arise from the fact that both the pressure and viscous components are related to the dynamic lift and trim moment in a non-linear way [3]. Therefore, the accurate prediction of resistance is linked to the accurate prediction of the running trim, sinkage and hence the lift force acting upon the hull. This is expanded upon by De Luca et. al. who state that the largest errors in resistance evaluation arise from errors in the dynamic trim [8]. Their statement is based upon observed errors in numerically calculated trim, and the relationship between dynamic trim and resistance put forward by [9], showing that the induced resistance component of total resistance varies with the dynamic trim angle. Additionally, as planing hulls are typically subject to small trim angles a small difference in the predicted trim will lead to large variations in the wetted surface, and as such, an incorrectly calculated trim angle will affect both the pressure and viscous components resistance components.

In order to predict the trim and resistance of a hull the pressure distribution and forces acting upon the hull must be calculated accurately. To do this it is vital that a fluid of the correct properties occupies the cells adjacent to the walls. A common problem in simulations of high-speed planing hulls that prevents this from happening is that of Numerical Ventilation (NV).

NUMERICAL VENTILATION

NV, or streaking, is a well-known problem when modelling planing hulls using the Volume of Fluid (VOF) model, however it is rarely discussed in depth by scientific papers [10], with some failing to mention the phenomenon altogether. It can be considered one of the main sources of error in numerical simulations of planing hulls [6], [8]. Böhm points out that the lack of discussion on this topic is attributable to the fact numerical ventilation only occurs with specific bodies, typically with a bow that forms a small, acute entrance angle with the free surface as is typical for yachts and high-performance vessels [11]. There is relative scarcity of ongoing research focusing on these hullforms when compared to conventional vessels, for which this problem does not occur. As such, there is limited discussion upon NV. It is noted that studies containing meaningful information on the topic do not often set out to discuss the issue, with the exception of Böhm's work [11], and as such the details contained within them are not always immediately apparent.

NV occurs when the free surface interface is not properly captured. Particles of air become trapped in the boundary layer in the first few cells nearest the wall and are transported under the hull. Olin reasons that this is introduced in the forward most spray area due to the fact that at some point along the hull, where the spray thickness approaches zero, the local cell size will be the same order of magnitude as the spray thickness [12]. The refinement in this area will not be sufficient to resolve the spray sheet, and as such no spray sheet will form forward of the

stagnation line. As opposed to forming a spray sheet, the information in these cells will be supplied under the hull and cause NV.

Another presented explanation focuses upon the interface capturing method that is employed by the simulation. One of the most common and successful advection schemes employed by CFD codes is the High-Resolution Interface Capturing (HRIC) scheme. This is based upon the Compressive Interface Capturing Scheme for Arbitrary Meshes and was introduced by [13] before being developed by [14]. Star CCM+ takes the standard HRIC scheme and blends it with the Upwind Differencing (UD) scheme based upon an upper and lower value of the local Courant Friedrichs Lewy (CFL) number. This blending is introduced to bring stability and robustness to the scheme, however it is noted that the interface is more 'smudged' when a blended scheme is employed. It is, however, recommended that for a steady state solution a pure HRIC scheme is implemented [15]. Further to this recommendation, Anderillion and Alessandrini showed the local CFL dependent scheme to cause a loss of sharpness in the free surface interface [16]. De Luca et al. propose the use of this blended scheme as the potential main cause of NV [8].

If NV occurs, it has a notable effect on the calculation of the vessel's frictional resistance [10], [12]. The CFD code will compute a lower value for the shear stress as this component is calculated using the velocity and viscosity of the elements in the boundary layer. If NV has occurred this cell may contain a mixed fluid, and thus the properties of this mixed fluid (which has a lower viscosity) will be used as opposed to those of water. [10] presents a detailed examination of this effect. It will also have an impact on the calculation of the pressure distribution and the trim of the hull. Any error introduced into the trim has a large impact on both the pressure and frictional components resistance, as previously discussed.

A number of strategies to minimize the problem of numerical ventilation have been proposed:

1. Viola et. al. found that using first order discretization for the convection terms lead an increase in NV [10]. It also found that using first order discretization lead to an increase in numerical diffusion & an increase in the computed resistance. The interface between water and air became less sharp and the transition occurred over a greater number of cells.
2. Viola et. al. also found that the timestep had an impact on whether NV was present or not [10]. As the timestep was increased so did the effects of NV.
3. Viola et. al. proposed a method to artificially suppress NV [10]. A source term is included in the transport equation for the air phase for the cells adjacent to the wall boundary layer. This removes the air mass from affected cells and replaces it with water. De Luca et. al. note that this strategy may introduce errors in the

conservation properties of mass and momentum [8], however [10] states that despite the violation of the continuity equation the effect on results is negligible.

4. Olin found that it was possible to reduce, but not suppress the NV through mesh refinement. The author states that a refined mesh close to the hull is not significantly reduce NV, however a free surface refinement upstream of the hull has a positive impact [12].
5. Böhm reasons that as simulations of towing tank procedures seek a steady state solution, the robustness of the HIRC scheme due to its local CFL dependency is not required. As such, it is possible to modify this scheme to remove the switch that blends it with the UD scheme, as it is known that the UD scheme leads to interface smearing [16]. Böhm found that this approach was well suited and gave a much sharper free surface interface, resulting in the minimization of numerical ventilation [11], as did De Luca et. al. [8]. It was also found that this approach had a positive impact on the calculated wave patterns due to the fact there was less interface smearing.

Böhm has performed the most extensive work on NV, comparing artificial suppression as suggested by [10] with both his modified HRIC scheme and the standard HRIC scheme [11]. He found artificial suppression to be the most successful method at removing NV, however advised caution as it introduces errors into the conservation properties of mass and momentum. It was also found that whilst not as impressive as artificial suppression, the modified HRIC scheme was far superior to the standard HRIC scheme.

Whilst the artificial suppression method is the most successful at eradicating NV, it should be noted that it is not always possible to utilise this approach. This is especially true when working with hulls for which air is purposefully introduced to the flow (such as stepped-hulls or air-lubricated hulls). In these cases, artificial suppression would be unable to differentiate between air accountable to NV and air that has been purposefully introduced. For these cases other methods must be investigated and a deeper understanding of the causation of NV developed.

AIMS

It is apparent that NV is a problem that simulations of planing hulls must overcome. There are a range of methods to minimise the effects, however there is no definitive solution to the problem and limited work in the public domain discussing it. Most studies that do include detailed discussion on the topic do not set out to investigate it, and so the information contained within them is not apparent when studying the literature in relation to NV. This paper sets out to bring together the available work, discussing the literature in a comprehensive manner. It aims to develop the understanding of the causes of NV, discussing these in detail so the findings may be easily implemented and applied to many cases. Novel solutions to help suppress NV are then explored,

based upon the determined root causes. The success and impact of different approaches are presented. This overall aim of this paper is to present a detailed discussion of the causes of NV, and to investigate and evaluate solutions to this, forming a guide to researchers who encounter NV over the course of their work.

METHODOLOGY

The study will use the published calm water experimental results of a series of high-speed hard chine planing hulls, generated by Taunton et. al. at Southampton University. For details on how these were generated please refer to [17]. Model C, shown in Figure 1, was selected at a speed of 9.21m/s as a benchmark case. The model was 2m in length, with a beam of 0.46m. This case was selected as it had a Froude number of 3.12, which is in line with the upper Froude numbers of similar studies investigating planing hulls through CFD.

FIGURE 1 - LINES PLAN OF MODEL C

Simulations will be set up using *CD Adapco's Star CCM+* CFD solver and run on the ARCHIE-WeST High Performance Computer, hosted by the University of Strathclyde.

This study primarily followed a qualitative methodology in its assessment of how successful a strategy was in minimizing NV. This assessment was made based upon the VOF plots of the hull. These VOF plots are presented in appendix A so that the readers may make their own judgment on the impact that a given method has. Following this a quantitative assessment was undertaken for the strategies identified as the most successful. No numerical evaluation metric was identified that could quantify the amount of NV that was present, so the resistance, trim and sinkage results were used as a measure of the accuracy of the simulation. As such the changes in results used in the quantitative assessment may not be solely accountable to the reduction in numerical ventilation, with some changes potentially being attributable to changes in the meshing strategy or other altered factors. Despite this, it provides a good measure of the level of accuracy of each case, and when combined with the qualitative assessment of the VOF plots an indisputable case may be made for the successful strategies.

NUMERICAL MODELING

This section will provide details upon the numerical simulation approaches utilised by this study, however it will not provide detailed information upon the numerical workings of the CFD code. Detailed information into the inner workings of CFD can be found in [18].

Physics Modelling

Larson et. al. state that the two-equation turbulence models have been shown to give accurate predictions in ship hydrodynamics [19]. The ITTC concluded from their analysis of the entries to

Gothenburg 2010 Workshop that there was no visible improvement in accuracy for resistance prediction when turbulence models that are more advanced than the two-equation models were used [1]. It found that $k - \omega$ was by far the most applied turbulence model with 80% of the submissions for the workshop using some form of variation of it. The ITTC also concluded that for resistance calculations the turbulence modelling has little effect on the prediction accuracy [2].

A review of other studies using CFD for planing hull performance prediction found that the majority of simulations use either $k - \varepsilon$ [8], [20]–[24] or $k - \omega$ SST [5]–[7], [25]–[27]. Whilst both models have been shown to be comparable in terms of resistance prediction the $k - \omega$ SST is known to be superior at predicting separating flows and wake patterns [1], [2]. As such, this model was selected despite the fact that it is more computationally expensive.

The Volume of Fluid (VOF) method was used to model and track the position of the free surface. This simple-multiphase model is well-suited for simulating flows of immiscible fluids and is known for its numerical efficiency. When the VOF model is used a new variable is introduced to define the spatial distribution of each phase at a given time. This is known as the volume fraction. A volume fraction of 0.5 represents a cell that contains 50% water and 50% air, and as such this is used to define the free surface. To help ensure that there was a sharp interface between the phases a second-order discretization scheme was used, as suggested by [15]. This agrees with the work of Viola et. al. who found a second-order convective discretization scheme to minimise NV and improve the accuracy of the simulation [10].

An all wall y^+ wall treatment was selected. This is a hybrid approach that emulates a low y^+ wall treatment for fine meshes ($y^+ > 1$) and the high y^+ wall treatment for course meshes ($y^+ < 30$). It is capable of producing reasonable answers for meshes of intermediate resolution ($1 < y^+ < 30$) through the use of a blending function.

An average y^+ of 40 was achieved on the wetted hull. This meant that for the wetted surface the viscous sublayer was not resolved and instead wall functions are used to obtain the boundary conditions of the continuum equations. The main advantage of the high y^+ wall treatment is that there are significant savings in computational time due to the reduction in the number of near-wall cells [15].

Timestep

The timestep can be selected either to satisfy the CFL condition or to resolve the flow features of interest. The ITTC recommend that for standard pseudo-transient resistance simulations a timestep that will satisfactorily resolve the flow features is a function of the vessels speed and the length of the hull, such that [28]:

$$\Delta t = 0.005 \sim 0.01 \frac{L}{U} \quad (1)$$

An extensive study was undertaken to determine the appropriate timestep. The results are visible in Figure 2. These are then employed in a formal Verification study to assess the levels of numerical uncertainty accountable to the temporal discretization later in the paper.

FIGURE 2 - TIMESTEP STUDY

This study found that a timestep that was coarser than the finest ITTC recommendation was suitably accurate. The selected timestep was 0.00304 and was determined using the following equation:

$$\Delta t = 0.02 \frac{L}{U} \quad (2)$$

The ITTC define L as the length between perpendiculars of the vessel. For the purpose of this study, L in the timestep calculation was taken to be the wetted length of the keel of the vessel.

The timestep study also showed that satisfying the CFL condition for all cells resulted in an unjustifiable increase in computational time with a negligible impact upon the results. While only the resistance results are shown, a timestep of 0.00044 produced results within 0.24% (for all measures) of the selected timestep of 0.00304. Satisfying the CFL condition also had negligible impact upon the levels of NV.

Computational Domain

It is well known that when using CFD the domain must be an appropriate size, with boundaries being placed sufficiently far from the hull to ensure they have no effect on the solution. The ITTC recommend that the inlet and exterior boundaries are located 1-2 L from the hull, with the outlet being placed 3-5 L downstream [19]. Care was taken to ensure that the wake of the hull would not intersect with the exterior boundary. The size of the computational domain was selected in accordance with the

ITTC recommendations [19] and can be seen in Figure 3. As is common practice in calm water marine resistance simulations the solution was assumed to be symmetrical with only half of the hull being modelled in conjunction with a symmetry condition on the center plane. This halves the computational demand over modelling the whole hull.

FIGURE 3 - BOUNDARY CONDITIONS AND DOMAIN SIZES

Boundary Conditions

In all CFD simulations, the selection of appropriate boundary conditions is vital for both the determination of an accurate solution and the prevention of unnecessary computational costs.

A study was carried out to determine the effects of boundary conditions on the level of NV. It was found that the selection of boundary conditions had no discernable impact upon the level of NV, however the impact upon accuracy and runtime for the various combinations of boundary conditions will be presented in this section. It should be noted that the boundary condition study was done using one of the preliminary meshes used to establish the final set up used for this study, however it remained constant for the duration of the boundary condition study.

It is commonplace in marine CFD to model the far field boundaries as free flow. A boundary may be considered far field if it is in deep water, and is over one ship length from the hull for Froude numbers over 0.2 [19]. Different authors implement this free flow condition at the far field boundaries through the use of inlet, slip wall and symmetry conditions. As such the investigation looked at the impact of using these boundary conditions. An additional condition of non-slip walls was also included to determine if it had an impact on the level of NV. This was studied using the ITTC domain size so that only one parameter was altered. While this condition was not physically representative of the model tests it did show that this selection had no impact on the level of NV. Implementing this non-slip condition increased the mesh count significantly as prism layer mesh was required on each wall surface. It is generally accepted that for far field boundaries this level of fidelity doesn't

significantly alter the results, and that modeling a free flow condition is a sufficiently accurate [19].

The boundary conditions combinations that were tested were:

Table 1. Grid Convergence Study.

Case	Side Boundary	Top Boundary	Bottom Boundary
Non-Slip	Non-Slip Wall	Inlet	Non-Slip Wall
Symmetry Inlet & Symmetry	Symmetry Symmetry	Symmetry Inlet	Symmetry Inlet
Slip Inlet	Slip Wall Inlet	Slip Wall Inlet	Slip Wall Inlet

FIGURE 4 – RESULTS OF THE BC STUDY

The tank test results of [17] were used as a reference when calculating percentage errors. In all cases modelling free flow at the boundary there a 0.041% variation in resistance error, 0.078% in trim, and 0.070% for sinkage. As mentioned, there was also no impact upon the numerical ventilation present in the simulation. The only discernable differences caused by the choice of boundary condition was the run time of the simulation. As such, the inlet condition was selected for use in all other simulations as it was the fastest.

The non-slip case showed a slightly larger variation in results than the free flow cases. It was discounted as it was not physically representative of the problem being modelled due to the domain being ITTC size, rather than the same as the test tank. As mentioned, the purpose of investigating this was to look at impact upon NV, however, a future study into the sensitivity of a planing hull simulation with regard to domain size when modeled with non-slip walls may be of interest.

The VOF Wave Damping option was utilised to apply a damping zone of 1.25L, as recommended by [29], to the side and outlet in

order to reduce wave reflections and the influence of the boundaries on the solution. This damping introduces a vertical resistance to vertical motion and suppresses waves.

A preliminary simulation was run with wave damping included and excluded. It was found to have minimal effects on the results (-0.016% error in resistance, 0.048% in trim and 0.012% in sinkage), which is confirmation that the boundaries were placed far enough from the hull for wave reflection to have minimal influence. Interestingly, it was found that the inclusion of wave damping reduced the runtime of the simulation by 1.77%.

Computational Grid

The Dynamic Fluid Body Interaction (DFBI) module allows a simulation to include the motion of a vessel in response to the shear and pressure forces exerted by the flow, and to any additional forces that are user defined. Star CCM+ calculates the force and moments that act upon the vessel before solving the governing equations of rigid body motion to determine the new position of the vessel. This model allows a body to have up to six degrees of motion, however, to simplify the simulation the vessel will only be free to move in two – pitch and heave. The equilibrium motion option is employed to achieve a quasi-steady-state equilibrium position of the vessel. This option means the body motion is calculated between longer intervals to reduce the time required to achieve a steady position [15].

The hydrodynamic field generated by a planing hull is far more complex than that of a conventional displacement hull with a small error in the predicted trim having a large impact upon the total resistance. There are a number of approaches that may be considered non-conventional for calm water resistance simulations that allow the mesh to change dynamically with the motion of the hull when implemented. These approaches help simulations maintain numerical accuracy while the hull is in inclined positions [30].

The most complex and computationally demanding of the approaches is the Chrimera Grid, or Overset Mesh. Overset Meshes typically involve a background mesh that is tailored to the environment, and one or more overset grids that are tailored and attached to the body, which overlaps with the background mesh. An overset mesh approach is very useful when dealing with moving bodies and Fluid Structure Interaction (FSI) as it offers far greater flexibility over standard meshing techniques. The approach's key advantage is that the grid system around the hull moves with the hulls motion. This means that the re-meshing or deformation of elements is not required, and the mesh remains consistent in terms of element quality. It is well known to be capable of modelling the large motions of a planing hull, and is recommended for configurations involving body-motion [19]. As such it was decided to implement an overset mesh approach.

There is no definitive recommendation made on the sizing of the overset zone. This zone should be large enough so that the flow features of interest remain within the overset domain and do not

have to pass through the doner cells to the background mesh. It is vital the the overlapping cells between the overset and background regions are of similar sizes to prevent interpolation errors as data is passed between them. The size of the overset domain was selected to be in line with similar studies. These studies all featured the same length of 1.5L, breadth ranging from 1.5B to 5B, and heights ranging from 2.5D to 6D [6], [8], [29], [31]. The overset region that was generated was 1.75L in length, 4B in width, and 4D in height. It can be seen in Figure 5, however due to the density of the mesh it is somewhat difficult to make out the bow, stern, and finest freesurface refinements.

FIGURE 5 - OVERSET REGION

The mesh was generated using the automated meshing capability of Star CCM+, which relies upon the Cartesian cut-cell method. The trimmed cell mesher presents a robust and efficient method of producing a high-quality grid, predominantly made up of unstructured hexahedral cells with polyhedral cells next to the surface. It constructs a template mesh from the target sizes and then trims this using the input surfaces. It allows for a large degree of control through the use of local surface and volumetric controls that allow the user to increase or decrease the mesh density. Growth parameters can also be used to ensure that there is a smooth transitioning of the mesh and prevent the introduction of numerical errors.

The mesh was set up with areas of progressively refined mesh to ensure each area of interest was sufficiently fine. Three layers of refinement were used for the free surface, the hull box and the wake region. Additional refinements were included for the bow, the stern and for the free surface upstream of the hull, as will be discussed later. The refinements can be seen in Figure 6. Care was taken to follow the overset guidelines as laid out by [15]. Of key importance was to ensure that cells in the overlapping region between the overset and background meshes are of similar sizes. This helps reduce any interpolation errors to be of the same order as other discretization errors.

FIGURE 6 – COMPUTATIONAL GRID

For the generation of the mesh a base size of 0.025m was selected. This was chosen as a function of the vessel length and was 1.25% L. This is in line with similar studies, for which base size ranged from 1.3% L to 4.7% L [5], [6], [15], [29]. All refinement zones were sized as a percentage of this base size. The sizing's for the final simulation were as follows:

Table 2. Grid Convergence Study.

Refinement Zone	X	Y	Z
Surface Mesh	50%	50%	50%
Hull Box [Near]	50%	50%	50%
Hull Box [Mid]	100%	100%	50%
Hull Box [Far]	200%	200%	100%
Overset Interface	200%	200%	100%
Wakebox [Near]	100%	100%	100%
Wakebox [Mid]	200%	200%	200%
Wakebox [Far]	400%	400%	400%
Freesurface [Upstream]	800%	800%	3.125%
Freesurface [Near]	800%	800%	12.5%
Freesurface [Mid]	800%	800%	25%
Freesurface [Far]	800%	800%	50%
Bow	6.25%	6.25%	6.25%
Stern	6.25%	6.25%	6.25%

When undertaking the mesh study the base size was changed so that the mesh strategy and relative sizes remained constant, whilst the sizes of the cells increased or decreased.

The prism layer mesher was used in conjunction with the trimmed cell mesher to generate orthogonal prismatic cells next to the hull. Utilising the prism layer mesher generates high-aspect ratio cells that are aligned with the flow next to the wall. This allows the software to resolve high velocity gradients that are associated with the boundary layer and increases the accuracy of the simulation. The initial thickness of the prism layer was calculated as the thickness of the turbulent flow over a flat plate, as given by:

$$\frac{\delta}{x} = 0.37R_n^{-\frac{1}{5}} \quad (3)$$

A stretching ratio of 1.2 as suggested by [19] was utilised, with a first wall cell height that was calculated to give a y^+ of 40. The thickness of each layer of the cells in the prism layer was calculated and the layer of a size that would naturally grow into the cell size of the volume mesh was chosen as the final prism layer. Whilst this meant that the prism layer thickness was 0.015m as opposed to 0.020m it ensured there was a far smoother transition in the mesh. Without this alteration the outer prism layer cells would have been larger than the volume mesh beside them.

For the purpose of investigating numerical ventilation a number of meshes were used ranging from 2.5M – 6M cells. The final mesh that was developed and will be used for the continuation of this work contained 6m cells. This mesh density was established through a mesh study as presented in the following section, utilised the final simulation developed as a result of this study to assess the associated levels of numerical uncertainty.

VERIFICATION

Before employing the simulations developed over the course of this study it is vital to undertake a comprehensive verification study so that the level of numerical uncertainty is understood, and the results may be used with confidence. While the aim of this paper was to investigate, in more general terms, strategies to minimise NV and the simulations are not used to produce further data, the verification study of the final simulation is included in the following section.

Guidelines on the best practice when performing a verification study in relation to a numerical marine analysis have been published by the ITTC [32]. An overview of this procedure will be given, however for further details please refer to the aforementioned document. Before continuing with this overview, it is first beneficial to define verification.

Verification is the quantitative assessment of the numerical uncertainty (U_{SN}) and when conditions permit, estimating the sign and magnitude of the numerical error (δ_{SN}^*) and the uncertainty (U_{SCN}) in that estimate. It is used to determine if a computational simulation accurately represents the conceptual model [33].

The recommendations made in [32] are based upon the work of [34], and follow an approach in which the errors and uncertainties are defined in a manner that is consistent with the uncertainty analysis of experimental work. The simulation error (δ_S) is defined as the difference between a simulation's result (S) and the truth (T), and is made up of modelling (δ_{SM}) and numerical (δ_{SN}) errors.

The analysis relies upon Richardson Extrapolation (RE), which forms the basis for existing quantitative numerical uncertainty and error estimates for both grid and timestep convergence [35]. The error is expanded in a power series, with integer powers of grid spacing or timestep taken as a finite sum. When it is assumed that the solutions lie within the asymptotic range it is acceptable that only the first term is considered, leading to a so-called triplet study.

The Correction Factor approach was employed, providing a quantitative measure of defining how far from a solution is from the asymptotic range, and then approximately accounting for the effects of higher order terms. This is based upon verification studies for 1D wave equations and 2D Laplace equation analytical benchmarks, which showed one-term RE error estimates to be poor when out with the asymptotic range, however that these could be improved by multiplying them by a

correction factor. Following this verification process allows the numerical uncertainty to be assessed.

Verification Case

Having developed the final simulation set up (Case 4 in table 4), a verification study was undertaken, with both grid and timestep studies being conducted. The grid and timestep studies employed a refinement ratio of $\sqrt{2}$ as suggested by [32].

The triplet studies for both mesh density and timestep produced results that displayed monatomic convergence. The specific uncertainties were calculated following the correction factor approach, as detailed in [32]. Prior to the undertaking of triplet studies, it was ensured that the iterative uncertainty was negligible and didn't contaminate the results. The calculated uncertainties are shown in table 3 & 4.

Table 3. Grid Convergence Study.

Parameter	r_G	EFD	S_1	S_2	S_3	R_G	U_G	U'_G
Cell Count	-	-	5,904,385	2,774,176	1,368,383	-	-	-
Resistance [N]	$\sqrt{2}$	69.98	76.675	77.013	77.639	0.539	0.510	0.665 %
Sinkage [m]	$\sqrt{2}$	0.05	0.05417	0.05419	0.05557	0.015	0.000042	0.077 %
Trim [Deg]	$\sqrt{2}$	1.73	2.066	2.064	2.017	0.173	0.003567	0.173 %

Table 4. Timestep Convergence Study.

Parameter	r_T	EFD	S_1	S_2	S_3	R_T	U_T	U'_T
Timestep [s]	-	-	3.04E-03	4.30E-03	6.08E-02	-	-	-
Resistance [N]	$\sqrt{2}$	69.98	76.675	76.775	77.079	0.331	0.151	0.197 %
Sinkage [m]	$\sqrt{2}$	0.05	0.05417	0.05415	0.05411	0.536	0.000033	0.060 %
Trim [Deg]	$\sqrt{2}$	1.73	2.066	2.064	2.062	0.871	0.031627	1.531 %

When the uncertainty accountable to both temporal and spatial discretization are combined the total numerical uncertainty for resistance is 0.20%, sinkage is 0.06% and trim is 1.53%. These values are all suitably small indicating that the simulation may be considered accurate in the sense that only small uncertainties are introduced from the choice of temporal and spatial discretization.

RESULTS & DISCUSSION

Following the initial run of the simulation, it was confirmed that it suffered from a significant amount of NV, and was incapable of producing accurate results as is visible in Figure 7. Through the course of the study the problem of NV was minimised from a level that made the simulation invalid, to a level at that was acceptable for engineering applications and had minimal effect on the results, as shown in Figure 10. Through a combination of the modified HRIC scheme and the correct mesh refinement, it was possible to minimise NV to two thin streaks containing 96-98% water.

FIGURE 7 - INADAQUATE MESH

FIGURE 8 - SIMULATION WITH STANDARD HRIC SCHEME

FIGURE 9 - SIMULATION WITH MODIFIED HRIC SCHEME

FIGURE 10 - SIMULATION WITH MODIFIED HRIC SCHEME & MESH REFINEMENT

FIGURE 11 - SIMULATION WITH MODIFIED HRIC SCHEME & MESH REFINEMENT ZOOMED IN (90% - 100% WATER)

The following section will discuss the potential solutions that were tested and their degree of success, as well as discussing the root causes of NV.

Modified HRIC Scheme

As was suggested by Böhm, and utilised by several other authors investigating high speed planing hulls through the use of CFD a modified HRIC scheme was employed [11]. Due to the removal of the CFL dependency and the blending with an UD scheme the ability of the simulation to capture the interface between the two phases was improved. This led to a significant reduction in the NV as well as giving the remaining NV a far sharper interface. This improvement can be seen as the change from Figure 8 to Figure 9.

Lowering CFL Number

The CFL number is the ratio of the time-step to the mesh convection time-scale. It essentially defines the number of cells that a particle of fluid will pass through in each timestep. It is recommended that the CFL number is less than or equal to 1 for numerical stability, however as a calm water resistance simulation is seeking a steady state solution larger CFL numbers give equally accurate results. It was theorized that a large CFL number at the point of hull entry would result in the code “losing track” of the air partials and introduce them into the flow under the hull due to the fact they were travelling through multiple cells in every timestep. A range of CFL numbers were tested and it was found to have little to no effect on NV. In the timestep study the smallest timestep had a CFL of around 12 at the point of hull entry, whilst the largest timestep had a CFL of around 100. There was no noticeable effect on the NV between the two.

As a final check the CFL was lowered to have a value of 0.5 at the point of hull entry, which required a timestep 20 times smaller than the ITTC formulation. This was found to reduce the percentage of air in the streaks from 3.5% to 2%, a small improvement, however it did not justify the extra computational time.

Boundary Conditions

As part of the BC study the NV was also checked. It was found that the choice of BC’s had no impact upon the NV that was present.

Domain Size

A smaller domain of the same dimensions as the towing tank in which the tests were originally carried out was also tested. This resulted in a domain that 2.375m wide and 1.68m deep. The distance from the hull to the inlet and outlet remained constant. The domain size was also found to have no impact upon the NV.

Turbulence Model

The $k - \epsilon$ turbulence model was tested, however it was found to have no impact upon the NV.

Sharpening Factor

The sharpening factor attempts to reduce numerical diffusion and improves the resolution of the interface between phases. It does this by introducing a new anti-diffusion velocity term into the VOF transport equation. A known problem with increasing the

sharpening factor is that it may result in a non-physical alignment of the free surface with the gridlines, which was found to result in a much flatter wake. A sharpening factor of 0.2 was included but it was found to be detrimental. Rather than sharpening the interface and preventing NV it was found to sharpen the interface of the NV under the hull, resulting in more clearly defined streaking.

Mesh Refinement

Mesh refinement was the best solution to the NV problem, after the modified HRIC scheme. The root cause of NV is when the free surface interface becomes blurred. The modified HRIC Scheme helps prevent this, which is accountable to its success. An inadequate mesh may also result in interface smearing through a number of causes.

The first cause of interface smearing accountable to the mesh arises from the prism layer. As was discussed earlier NV only occurs for specific bodies, typically with a bow that forms a small, acute entrance angle with the free surface. When bodies such as these are meshed the prism layer mesh and the volume mesh that have a small angle between them, as seen in Figure 12. In the case of a conventional ship this angle would be large, possibly even 90 degrees. Due to this the cells in the prism layer mesh are not aligned with the flow. It is well known that when the free surface is not aligned with the mesh numerical diffusion will occur. This results in interface smearing at the point of entry of the hull, as seen in Figure 12. It can also be seen that air is transported under the hull in the near wall cells, resulting in NV.

FIGURE 12 - PRISM LAYER MESH

Any meshing strategy that prevented interface smearing in the prism layer resulted in a vast improvement on NV. The number of prism layers had a large effect. It is normally advisable to have the last cell of the prism layer and the first cell of volume mesh of comparable sizes, however implementing this was found to be detrimental. As can be seen in Figure 12 the last layer of the prism layer appears to be too large, however adding more prism layers was found to result in further interface smearing. The thickness of the prism layer also had a large effect on NV. A thick prism layer meant that the free surface had a considerably larger zone in which interface smearing occurred as there were more cells that were misaligned with the flow. A strategy to minimise this was developed, in which the prism layer thickness was reduced to 0.25 of the original value at the point of hull entry, as seen in Figure 13. This was found to significantly reduce NV that

was introduced due to interface blurring as a result of the prism layer cells.

FIGURE 13 - COLLAPSING PRISM LAYER

The second cause of free surface blurring arose from inadequate mesh refinements. The first refinement strategy was to increase the resolution of the mesh in the bow area. This ensures that the mesh is capable of resolving the thin spray root that forms and prevents air being dragged under the hull. The bow refinement on its own however, was found to be detrimental to the NV. This was because the bow refinement had a finer z-refinement than the free surface refinement, which resulted in the interface smearing when the free surface entered the bow-refinement zone; the free surface that was modelled by one cell was modelled by multiple cells, as seen in Figure 14. It is therefore necessary to also include an upstream free surface refinement zone of equal z-refinement to that of the bow-refinement. This is in agreement with what was found by Olin, who stated that upstream refinement had the largest positive impact [12].

FIGURE 14 - INTERFACE SMEARING DUE TO DOWNSTREAM Z-REFINEMENT

Adding upstream free surface refinement had a larger effect than the bow refinement. This is accountable to the way in which Star CCM+ generates the mesh. When an upstream refinement added, the refinement is projected through the prism layer mesh onto the hull surface mesh as well in the zone where the upstream refinement meets the hull. This means that when a free surface refinement is implemented it essentially adds a surface refinement to the hull in the area in which it intersects with the hull. Adding a bow refinement in addition to the upstream refinement helps ensure that the refined area is sufficiently large to capture the flow characteristics in the entry region. Having a bow refinement was found to further minimise NV over an upstream free surface refinement alone.

When the levels of refinement were varied for each of the two zones it was found to have a large effect on the NV. The level of bow refinement had less impact as the free surface refinement projects onto the hull at the intersection zone, but it still showed some effect. The level of free surface refinement had a notable effect on the NV. There appears to be a ‘sweet spot’ for the level of both refinements that was found by making systematic variations. If they are more or less refined than this ‘sweet spot’ then the NV becomes worse. Of interest is that this ‘sweet spot’ is relative to the rest of the mesh, rather than absolute sizes. This was noted during a mesh study when the sizing of the entire mesh was altered but the level of NV remained constant despite the fact that the absolute size within the bow and upstream refinements had changed. Previously changing the absolute sizes in these zones and leaving the mesh constant had resulting in increased NV.

During the preliminary set up of the simulation a low y^+ wall treatment was initially attempted. This set up had a combination of a large number of prism layers and an inadequate free surface mesh in which the bow z-refinement was finer than the free surface z-refinement. The loss of the water air interface due to this set up is seen in Figure 15. It should be noted that a modified HRIC has not been implemented in this set up. This is a worst-case scenario in which the potential to cause large amounts of NV are displayed. The same simulation was set up with a high y^+ wall treatment, the only difference being the number of prism layers. This too suffered from a large amount of NV, however it was considerably less than for the low y^+ case, reinforces the idea that reducing the number of prism layers plays a large part in preventing NV. The effect of high y^+ case of this inadequate set up on the VOF plot of the hull is shown in Figure 7.

FIGURE 15 - LOW y^+ WALL TREATMENT INTERFACE SMEARING

Whilst altering the mesh a second source of NV was discovered further aft of the bow entry point. This resulted in two additional streaks as seen in Figure 16.

FIGURE 16 - SECOND SOURCE OF NV

This was once again the result of interface smearing where the volume mesh met the prism layer. In this case, the volume mesh cells were considerably larger than the prism layer, which lead to smearing as seen in Figure 17. As the slice used to generate the scene was systematically moved aft the interface smearing could be seen as it was developed into a bubble that was drawn into the prism layer. Here it moved inwards until it smeared on the hull. The solution to this second source was to ensure that the volume and prism layer mesh were of comparable sizes until the free surface met the chine. In a practical sense, this was resolved by extending the upstream free surface refinement aft until it met the chine.

FIGURE 17 - INTERFACE SMEARING AT SECOND SOURCE OF NV

Quantitative Analysis

In addition to the qualitative assessment that has been presented so far it is possible to carry out a quantitative assessment for the four main cases as presented in Figure 7 to Figure 10. The changes implemented in the progression of these figures are the key strategies to ensuring numerical ventilation is reduced. A brief summary of each of the figures is given before the results are presented.

- Figure 7 (Case 1) has an inadequate mesh set up in which there is a finer z-refinement in the bow refinement zone than the upstream free surface z-refinement. This causes the free surface to be spread over several cells, which then suffers from a large degree of numerical diffusion in the prism layer as seen in Figure 15. (Note that Figure 15 is a low y^+ case, and Figure 7 is a high y^+ case, however the same phenomena is occurring.

- Figure 8 (Case 2) is the same as Figure 7, however aforementioned problems with the mesh were been corrected so that the simulation may be considered adequate.
- Figure 9 (Case 3) is the same as Figure 8, however a modified HRIC scheme has been implemented.
- Figure 10 (Case 4) is the same as Figure 9, however the prism layer has been collapsed at the point of entry and suitable mesh refinements have been added in the bow and upstream free surface regions.

Table 5. Quantitative Results [Absolute Values]

Case	Resistance [N]	Trim [Deg]	Sinkage [m]
Case 1	53.19	2.141	0.0563
Case 2	75.53	2.190	0.0545
Case 3	76.17	2.068	0.0544
Case 4	76.67	2.066	0.0542

Table 6. Quantitative Results [Error Values]

Case	Resistance [N]	Trim [Deg]	Sinkage [m]
Case 1	23.99%	-23.78%	12.70%
Case 2	-7.93%	-26.57%	9.09%
Case 3	-8.85%	-19.54%	8.83%
Case 4	-9.57%	-19.41%	8.34%

As can be seen the results for Case 1 contain significant error when compared to the experimental data, confirming that this simulation is inadequate and is incapable of producing reliable results due to the prevalence of NV.

When the mesh is corrected so that the free surface interface is no longer spread over several cells in Case 2 the amount of NV is seen to reduce dramatically. This causes the resistance results improve significantly. This case is now far more physically representative of what is happening in real life as there is no longer a mixture of air and water under the hull.

From Case 2 – Case 4 the changes are far smaller than the jump from Case 1 – 2. The resistance is seen to increase as the amount of NV reduces. This is to be expected as the fluid properties of the in the near wall cells will change as they contain less air, resulting in an increase in frictional resistance. While the error in comparison to the experimental results is seen to grow as the NV is reduced this should not be considered a negative thing. The NV is not physically there in the real-world case, and as such the simulations where it is eradicated are far more representative of this. While the simulations with NV present appear to be more accurate in terms of resistance this is only due to an error that is introduced into the simulation. More accurate methods of turbulence modelling or other changes to the simulation may improve the accuracy further, however it is always desirable to minimise NV as much as possible.

The trim and sinkage are both seen to progressively improve in accuracy as the NV is reduced. This is because the forces acting upon the hull and the pressure distributions are more accurately modeled in the cases where there is no mixture of fluids on the hull.

One of the key requirements when setting up CFD simulations is to sufficiently balance computational cost with simulation accuracy. This is a judgment that must be made on a case-by-case basis. Following this the uncertainty must be assessed though a verification study. There was a 1.61% increase in when the modified HRIC scheme was employed. This is computationally inexpensive method of improving the simulations accuracy. It is recommended that this approach be followed regardless of the availability of computational resources. When the prism layer was collapsed at the point of entry, and additional bow and upstream freesurface refinements were included 500k cells were added to the mesh, which lead to an increase of 9.25% in run time. As this work was performed on a HPC this led to an increase of 47 minutes, which was not considered significant and therefore the merit in accuracy was considered worth the

Summary

The effectiveness of each of the investigated methods has been summarized and is presented in Table 1. They have been ranked in effectiveness from 1 – 3 with one being very effective and three being ineffective. The strategies deemed to be effective have been confirmed through both qualitative assessment of their results, and quantitative assessment of the VOF plots. There was no quantitative assessment undertaken for strategies deemed to be ineffective based upon qualitative analysis of their VOF plots. These VOF plots are presented in Appendix A so that the reader may make their own judgment as to the effectiveness of each strategy. It should be noted that a strategy being classified as effective does not mean it will be sufficient on its own and that multiple methods may need to be employed to minimise the NV. It should also be noted that strategies classified as ineffective may be effective with further investigation, however they were not found to be in the scope of this study.

Table 7. Summary of Investigated Strategies

Strategy	Effectiveness
Modified HRIC Scheme	1
Lowering CFL Number	3
Boundary Conditions	3
Domain Size	3
Turbulence Model	3
Sharpening Factor	3
Mesh Refinement – Bow Refinement	2
Mesh Refinement – Upstream Free Surface Refinement	1
Mesh Refinement – Thickness of Prism Layer at Free Surface	1
Mesh Refinement – Number of Prism Layers	1
Mesh Refinement – Prism Layer to Volume Mesh Size Difference	1

CONCLUSION

This study undertook a systematic study to establish strategies of minimising the levels of NV in a CFD simulation of a planing hull. Both qualitative and quantitative analysis were used to establish the strategies that were the most effective, giving confidence to the conclusions. Additionally, a complete verification study was undertaken to ensure that the numerical uncertainty associated with the temporal and spatial discretization of the simulations was understood. This further enhances the confidence in the results.

The main cause of NV identified in this study is when interface smearing occurs due to the simulation being incapable of tracking the free surface. This may be accountable to either the interface capturing scheme or the mesh. When the mesh was at fault, NV was introduced to the hull from two different sources:

- at the point of entry
- as streaks nearer the chine

When a more detailed understanding of the problem was developed and both sources were investigated, it was found that they both originated from the same cause – the interface between the prism layer mesh and the volume mesh. This is primarily accountable to the fact that the prism layer cells are not aligned with the free surface, which results in numerical diffusion. Whilst there is no way to avoid this a number of solutions were tested. It was found that whilst it is not possible to eradicate NV it is possible to reduce it to a level at which it will have little to no bearing on the results and is acceptable for engineering applications. Through the course of this study the NV present in the simulation was reduced from two 0.055m wide streaks with 90% air content, to two 0.011m wide streaks of 4% to 2% air content.

Previous work that presented solutions to NV was compiled and the applicability of each was tested. It was found that Bohm's modified HRIC [11] and Olin's upstream refinements [12] were the most capable, however from the literature it is suggested to use Viola et. Al's. artificial suppression where applicable [10]. It was found that the timestep and the CFL of the free surface had little impact when trying to reduce NV. Additionally, a number of other solutions were tried to help gain a better understanding of NV and to help future researchers save time by establishing what is and is not successful.

It was found that reducing the level of numerical ventilation in this case was slightly detrimental to the resistance results. The simulation was found to over predict resistance and when the NV was removed through the implementation of the proposed strategies the fluid properties in the near wall cells changed, and the frictional resistance increased. While this increased the error in the resistance results this was in fact more physically representative of the real-world case and it is necessary to reduce it as much as possible in all simulations. Further enhancements to the simulation through the changes to the turbulence approach may further improve the accuracy. Reducing the NV was found to improve the trim and sinkage results as the pressures and forces acting upon the hull were calculated more accurately.

A novel solution of reducing the thickness of the prism layer at the point of water entry was developed and deemed successful. Through the investigation into the effects of mesh parameters on NV and by laying out clearly how interface smearing may be introduced through an inadequate mesh it is hoped that the understanding of the NV will be further developed. A more detailed and widespread understanding of NV will hopefully increase the accuracy of high-speed planing hull simulations and by minimising the problem it will ensure that all future work is more reliable. Future work in the area should investigate the interface capturing scheme settings in more detail. Whilst it is possible to minimise Numerical Ventilation through the mesh refinement it may not be possible to eradicate it fully using this approach.

ACKNOWLEDGMENTS

Results were obtained using the ARCHIE-WeSt High Performance Computer (www.archie-west.ac.uk) based at the University of Strathclyde.

APPENDIX A – VOF PLOTS

The VOF plots for the various strategies are shown here. It should be noted that in each comparison only the factor being studied was changed.

Modified HRIC Scheme

The first image shows the NV for the HRIC scheme that is implemented as standard, where it is blended with UD. The second has had the blending with UD removed.

Lowering CFL Number

The first image has a timestep of 0.00608. The second has had this lowered to 0.00044 to reduce the CFL to 1 in all cells.

Boundary Conditions

Non-Slip

Symmetry

Inlet & Symmetry

Slip

Inlet

Domain Size
ITTC size

Sharpening Factor
No Sharpening Factor

Test Tank Size

0.2 sharpening factor included

Turbulence Model
k-w

Mesh Refinements
Best combination of mesh refinements

k-e

Unsuitable combinations of mesh refinement levels

REFERENCES

- [1] L. Larson, F. Stern, and M. Visonneau, *Numerical Ship Hydrodynamics - An Assessment of the Gothenburg 2010 Workshop*. 2014.
- [2] ITTC Specialist Committee on CFD in Marine Hydrodynamics, "Specialist Committee on CFD in Marine Hydrodynamics Final report and Recommendations to the 27 th ITTC," 2014. Accessed: Dec. 11, 2018. [Online]. Available: <https://itc.info/media/6097/sc-cfd.pdf>.
- [3] S. Brizzolara and F. Serra, "Accuracy of CFD codes in the prediction of planing surfaces hydrodynamic characteristics," *2nd Int. Conf. Mar. Res. Transp.*, no. June 2007, pp. 147–158, 2007, [Online]. Available: <http://www.icmrt07.unina.it/Proceedings/Papers/B/14.pdf>.
- [4] S. T. G. Veysi, M. Bakhtiari, H. Ghassemi, and M. Ghiasi, "Toward numerical modeling of the stepped and non-stepped planing hull," *J. Brazilian Soc. Mech. Sci. Eng.*, vol. 37, no. 6, pp. 1635–1645, Nov. 2015, doi: 10.1007/s40430-014-0266-4.
- [5] S. Mancini, F. De Luca, and A. Ramolini, "Towards CFD guidelines for planing hull simulations based on the Naples Systematic Series," in *7th International Conference on Computational Methods in Marine Engineering, MARINE 2017*, 2017, vol. 2017-May, no. May.
- [6] A. De Marco, S. Mancini, S. Miranda, R. Scognamiglio, and L. Vitiello, "Experimental and numerical hydrodynamic analysis of a stepped planing hull," *Appl. Ocean Res.*, vol. 64, no. March, pp. 135–154, Mar. 2017, doi: 10.1016/j.apor.2017.02.004.
- [7] D. Frisk and L. Tegehall, "Prediction of High-Speed Planing Hull Resistance and Running Attitude. A Numerical Study Using Computational Fluid Dynamics," Gothenburg, 2015.
- [8] F. De Luca, S. Mancini, S. Miranda, and C. Pensa, "An Extended Verification and Validation Study of CFD Simulations for Planing Hulls," *J. Sh. Res.*, vol. 60, no. 2, pp. 101–118, 2016, doi: 10.5957/JOSR.60.2.160010.
- [9] W. Sottorf, "Experiments with planing surfaces," *Natl. Advis. Comm. Aeronaut. Tech. Memo. 661*, no. March, Mar. 1934, Accessed: Jan. 03, 2019. [Online]. Available: <https://ntrs.nasa.gov/search.jsp?R=19930094677>.
- [10] I. M. Viola, R. G. J. Flay, and R. Ponzini, "CFD analysis of the hydrodynamic performance of two candidate America's cup AC33 hulls," *Int. J. Small Cr. Technol.*, vol. 154, no. 1, pp. 1–12, 2012, doi: 10.3940/rina.ijsc.2012.b1.113.
- [11] C. BÖHM, "A Velocity Prediction Procedure for Sailing Yachts With a Hydrodynamicmodel Based on Integrated Fully Coupled Ranse-Free-Surface Simulations," Delft University of Technology, 2011.
- [12] Linus Olin, "Numerical modeling of spray sheet deflection on planing hulls," KTH Royal Institute of Technology, Stockholm, 2015.
- [13] O. Ubbink, "Numerical prediction of two fluid systems with sharp interfaces," University of London and Diploma of Imperial College, 1997.
- [14] S. MUZAFERIJA and M. PERIC, "Computation of free surface flows using interface-tracking and interface capturing methods," in *Nonlinear Water Wave Interaction*, Southampton: WIT Press, 1999.
- [15] CD Adapco, "Simcenter STAR-CCM+ Documentation." 2018, Accessed: Jan. 07, 2019. [Online]. Available: https://documentation.thesteveportal.plm.automation.siemens.com/starccmplus_latest_en/index.html#page/connect%2Fsplash.html.
- [16] Y. Andrillon and B. Alessandrini, "A 2D+T VOF Fully Coupled Formulation for Calculation of Breaking Free Surface Flow," in *Twenty-Fourth Symposium on Naval Hydrodynamics*, Washington, D.C.: National Academies Press, 2003.
- [17] D. J. Taunton, D. A. Hudson, and R. A. Sheno, "Characteristics of a series of high speed hard chine planing hulls – Part 1: Performance in calm water," *Int. J. Small Cr. Technol.*, vol. 152, pp. 55–75, 2010, doi: 10.3940/rina.ijsc.2011.b1.97.
- [18] J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*, Thrid. Berlin: Springer, 2002.
- [19] ITTC - Recommended Procedures and Guidelines, "Practical Guidelines for Ship CFD," 2011. Accessed: Jan. 03, 2019. [Online]. Available: <https://itc.info/media/1357/75-03-02-03.pdf>.
- [20] S. Brizzolara and D. Villa, "CFD SIMULATION OF

- PLANING HULLS,” in *Seventh International Conference On High-Performance Marine Vehicles*, 2010, Accessed: Oct. 23, 2018. [Online]. Available: https://s3.amazonaws.com/academia.edu.documents/32730317/Brizzolara_CFD_Planing-HullsV5.pdf?AWSAccessKeyId=AKIAIWOWYYGZ2Y53UL3A&Expires=1540308488&Signature=V%2FrE4Tvozubk8fc1RQ%2B%2FB54TLTw%3D&response-content-disposition=inline%3Bfilename%3DCFD_Simulat.
- [21] P. Lotfi, M. Ashrafizaadeh, and R. K. Esfahan, “Numerical investigation of a stepped planing hull in calm water,” *Ocean Eng.*, vol. 94, pp. 103–110, 2015, doi: 10.1016/j.oceaneng.2014.11.022.
- [22] M. Bakhtiari, S. Veysi, and H. Ghassemi, “Numerical Modeling of the Stepped Planing Hull in Calm Water,” *Int. J. Eng.*, vol. 29, no. 2, Feb. 2016, doi: 10.5829/idosi.ije.2016.29.02b.13.
- [23] O. F. Sukas, O. K. Kinaci, F. Cakici, and M. K. Gokce, “Hydrodynamic assessment of planing hulls using overset grids,” *Appl. Ocean Res.*, vol. 65, pp. 35–46, 2017, doi: 10.1016/j.apor.2017.03.015.
- [24] A. Dashtimanesh, A. Esfandiari, and S. Mancini, “Performance Prediction of Two-Stepped Planing Hulls Using Morphing Mesh Approach,” *J. Sh. Prod. Des.*, vol. 00, no. February 2018, pp. 1–13, 2017, doi: 10.5957/JSPD.160046.
- [25] H. Ghassemi, M. Kamarlouei, and S. T. G. Veysi, “A Hydrodynamic Methodology and CFD Analysis for Performance Prediction of Stepped Planing Hulls,” *Polish Marit. Res.*, vol. 22, no. 2, pp. 23–31, 2015, doi: 10.1515/pomr-2015-0014.
- [26] S. Wang, Y. Su, X. Zhang, and J. Yang, “RANSE simulation of high-speed planning craft in regular waves,” *J. Mar. Sci. Appl.*, vol. 11, no. 4, pp. 447–452, 2012, doi: 10.1007/s11804-012-1154-x.
- [27] T. Castiglione, F. Stern, S. Bova, and M. Kandasamy, “Numerical investigation of the seakeeping behavior of a catamaran advancing in regular head waves,” *Ocean Eng.*, vol. 38, no. 16, pp. 1806–1822, 2011, doi: 10.1016/j.oceaneng.2011.09.003.
- [28] 26th ITTC Specialist Committee on CFD in Marine Hydrodynamics, “Practical Guidelines for Ship CFD Applications,” 2014. doi: 10.1016/j.tca.2008.12.030.
- [29] T. Tezdogan, Y. K. Demirel, P. Kellett, M. Khorasanchi, A. Incecik, and O. Turan, “Full-scale unsteady RANS CFD simulations of ship behaviour and performance in head seas due to slow steaming,” *Ocean Eng.*, vol. 97, pp. 186–206, 2015, doi: 10.1016/j.oceaneng.2015.01.011.
- [30] E. Begovic, C. Bertorello, and S. Mancini, “Hydrodynamic performances of small size swath craft,” *Brodogradnja/Shipbuilding*, vol. 66, no. 4, pp. 1–22, 2015.
- [31] Ö. F. Sukas and M. K. Gökçe, “Prediction of Hydrodynamic Aspects of a Catamaran Using Overset Grid Design,” in *Thirteenth International Conference on Marine Sciences and Technologies Black Sea*, 2016.
- [32] Resistance Committee of 25th ITTC, “Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures,” 2008. [Online]. Available: <http://itc.info/media/4184/75-03-01-01.pdf>.
- [33] AIAA and A. I. of A. and, “Guide for the Verification and Validation of Computational Fluid Dynamics Simulations,” American Institute of Aeronautics and Astronautics, Inc., Washington, DC, Jan. 1998. doi: 10.2514/4.472855.
- [34] F. Stern, R. Wilson, and J. Shao, “Quantitative V&V of CFD simulations and certification of CFD codes,” *Int. J. Numer. Methods Fluids*, vol. 50, no. 11, pp. 1335–1355, 2006, doi: 10.1002/fld.1090.
- [35] T. Xing and F. Stern, “Factors of safety for Richardson extrapolation,” *J. Fluids Eng. Trans. ASME*, vol. 132, no. 6, pp. 0614031–0640313, Jun. 2010, doi: 10.1115/1.4001771.