

CFD Benchmark of FOAMSTAR: validation of a CFD tool

Auteur : Kyaw Swa Hein,

Promoteur(s) : 14999

Faculté : Faculté des Sciences appliquées

Diplôme : Master : ingénieur civil mécanicien, à finalité spécialisée en "Advanced Ship Design"

Année académique : 2020-2021

URI/URL : <http://hdl.handle.net/2268.2/13303>

Avertissement à l'attention des usagers :

Tous les documents placés en accès ouvert sur le site le site MatheO sont protégés par le droit d'auteur. Conformément aux principes énoncés par la "Budapest Open Access Initiative"(BOAI, 2002), l'utilisateur du site peut lire, télécharger, copier, transmettre, imprimer, chercher ou faire un lien vers le texte intégral de ces documents, les disséquer pour les indexer, s'en servir de données pour un logiciel, ou s'en servir à toute autre fin légale (ou prévue par la réglementation relative au droit d'auteur). Toute utilisation du document à des fins commerciales est strictement interdite.

Par ailleurs, l'utilisateur s'engage à respecter les droits moraux de l'auteur, principalement le droit à l'intégrité de l'oeuvre et le droit de paternité et ce dans toute utilisation que l'utilisateur entreprend. Ainsi, à titre d'exemple, lorsqu'il reproduira un document par extrait ou dans son intégralité, l'utilisateur citera de manière complète les sources telles que mentionnées ci-dessus. Toute utilisation non explicitement autorisée ci-avant (telle que par exemple, la modification du document ou son résumé) nécessite l'autorisation préalable et expresse des auteurs ou de leurs ayants droit.



Universität
Rostock



Traditio et Innovatio



SOLENT
UNIVERSITY
SOUTHAMPTON



Zachodniopomorski
Uniwersytet
Technologiczny
w Szczecinie



With the support of the
Erasmus+ Programme
of the European Union



BUREAU
VERITAS
SOLUTIONS
Marine & Offshore

CFD Benchmark of *foamStar*. validation of a CFD tool

Master Thesis

(Public Version)

submitted on 25 August 2021

by

KYAW SWA HEIN | 44300 NANTES | kyaw-swa-hein@eleves.ec-nantes.fr

Student ID No.: 200521Q (École Centrale de Nantes); s193329 (University of Liège)

Supervisors: Sébastien LOUBEYRE, Bureau Veritas Solutions

Théo DESCAMPS, Bureau Veritas Solutions

Reviewers: Zhe LI, École Centrale de Nantes

Guillaume DUCROZET, École Centrale de Nantes

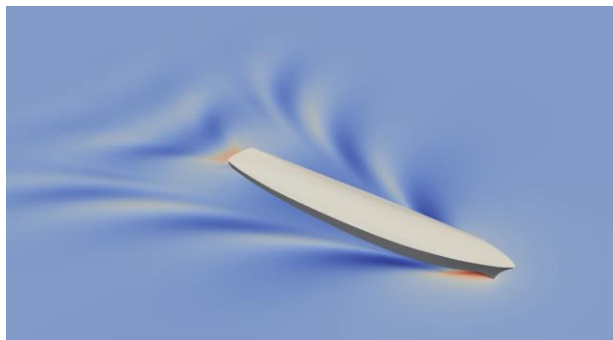
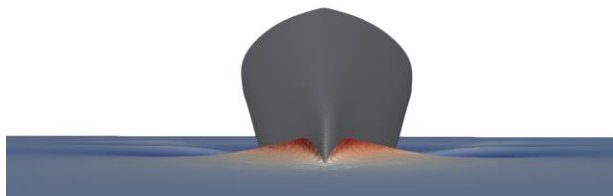




Internship Report

25/08/2021

CFD Benchmark of *foamStar* : validation of a CFD tool



**BUREAU
VERITAS**

SOLUTIONS
Marine & Offshore

PROJECT INFORMATION

Document Reference	HEIN KYAW SWA 2021 <i>foamStar</i>
Type of document	Internship Report
Project Name	CFD Benchmark of <i>foamStar</i> : validation of a CFD tool
# of pages & appendix	40

ISSUED BY

Company name	Bureau Veritas Solutions (BVS)
Department	Marine and Offshore – Fluid Dynamics
Address	4 Rue Duguay Trouin 44800 Saint-Herblain
Phone	+33 (0)9 69 39 10 09

ISSUED TO

Company name	Ecole Centrale de Nantes (ECN)
Address	1 rue de la Noe – BP 92101 F 44321 Nantes Cedex 3
Phone	+33 (0)2 40 37 16 00
E-mail	lionel.gentaz@ec-nantes.fr

VALIDATION PROCESS

	Authors	Verification	
Name	HEIN KYAW SWA	THEO DESCAMPS	SEBASTIEN LOUBEYRE
Function	Intern (Master Thesis)	PhD Candidate	Fluid Dynamics Engineer
Date	19/08/2021	20/08/2021	20/08/2021

DECLARATION OF AUTHORSHIP

I declare that this thesis and the work presented in it are my own and have been generated by me as the result of my own original research.

Where I have consulted the published work of others, this is always clearly attributed.

Where I have quoted from the work of others, the source is always given. With the exception of such quotations, this thesis is entirely my own work.

I have acknowledged all main sources of help.

Where the thesis is based on work done by myself jointly with others, I have made clear exactly what was done by others and what I have contributed myself.

This thesis contains no material that has been submitted previously, in whole or in part, for the award of any other academic degree or diploma.

I cede copyright of the thesis in favour of Bureau Veritas Solutions (BVS).

Date: 20th August, 2021

Signature: KYAW SWA HEIN

ACKNOWLEDGEMENTS

First, I would like to express my gratitude to EMShip+ program coordinator Prof. Philippe Rigo for offering me the opportunity to continue my Master's study in this advanced ship design program and for delivering lectures on ship hull structures.

I would like to thank Prof. André Hage, the professors from University of Liège (ULG) and other co-ordinating universities for providing lectures on practical ship design and supportive engineering lectures.

I am really grateful to the professors from École Centrale de Nantes (ECN) for their invaluable lectures on marine hydrodynamics - Prof. Pierre Ferrant in wave structure interaction, Prof. Guillaume Ducrozet in wave modelling, Prof. Lionel Gentaz and Prof. Zhe Li in numerical hydrodynamics, Prof. David Le Touzé in turbulence modelling and Prof. Félicien Bonnefoy in experimental hydrodynamics.

I would like to extend my thankfulness to those from Bureau Veritas Solutions Marine and Offshore (BVS) – Fluid Dynamics Department: Pierrick Sergent, Sébastien Loubeyre, Théo Descamps and Adèle Lesage for supervising me throughout my thesis internship period.

I wish to appreciate the great love from my grandma and two sisters.

Finally, it is my pleasure to work with my colleagues from EMShip+, all of those from ANAST, University of Liège, École Centrale de Nantes, Bureau Veritas and Bureau Veritas Solutions.

ABSTRACT

In predicting the resistance of ship hulls, experiments in towing tanks have been mostly performed before the extensive use of CFD applications. Time and cost turn out to be key factors in consideration along with the accuracy of results. High costs of experiments and limitations in testing several scenarios in towing tests led to an increased use of CFD codes for ship resistance and propulsion applications. Bureau Veritas (BV) Solutions has interests in developing in-house CFD codes in order to reduce license costs and to enable the engineers to contribute to the code development. One of these CFD tool is being jointly developed by Bureau Veritas (BV) and École Centrale de Nantes (ECN). It is based on OpenFOAM open-source libraries for obvious reasons of accessibility and cost. Its name is *foamStar*.

The work performed in this internship enabled to verify the robustness and sensitivity of *foamStar* in view of its industrial use on resistance cases. At first, multiple sensitivity studies both on numerical parameters and grid parameters were conducted. It showed the importance of good practices to optimise its use and highlighted some of the key setup options to be used. Moreover, some unstable cases/configurations were encountered showing the poor stability of *foamStar*, especially with regard to the mesh.

Then, the benchmark cases of KCS and DTMB 5415 hulls have been performed in calm water resistance computations. Good agreement was observed in resistance coefficients, free surface pattern and wave profiles between *foamStar* and experimental results. It can be said that *foamStar* can capture flow phenomena with the certain degree of accuracy. These results are encouraging and tend to validate the capability of the software for resistance predictions with a small tendency to over-estimate it.

For industrialisation purpose, the setup procedure to run *foamStar* calculations have also been automatized by modifying the BVS's existing script. The workability and the reliability is well established by testing it on hull optimisation cases. In addition, the results obtained for this optimisation study showed that while over-estimating the resistance, *foamStar* results are in line with ISIS-CFD results with regard to relative comparison of the various hull forms tested. It is a positive outcome which tends to validate its use for such applications. However, it needs to be investigated further prior to any industrial use as one case is not enough for validation.

Keywords: Hydrodynamics, Ship Resistance, *foamStar*, OpenFOAM, CFD Benchmark, Hull Optimisation, BVS, ECN

TABLE OF CONTENTS

1 /	<i>Introduction</i>	9
1.1/	foamStar Description	9
1.2/	Purpose of Internship.....	9
2 /	<i>Literature Review</i>	11
2.1/	General Introduction on OpenFOAM/foamStar	11
2.1.1/	OpenFOAM Description.....	11
2.1.2/	Standard and Extended Solvers for Marine Applications	11
2.1.3/	foamStar and Its Specifications	12
2.2/	Technical Review of Key OpenFOAM/foamStar Aspects	14
2.2.1/	Mathematical Introduction to Finite Volume Method in CFD	14
2.2.2/	Turbulence Models	14
2.2.3/	Finite Volume Discretisation	15
2.2.4/	Linear Solver Algorithms.....	15
2.3/	Basic Case Architecture in OpenFOAM – Focus on Utilities	17
2.3.1/	Mesh Generation Utilities.....	18
2.3.2/	Finite Volume Discretization Schemes ('fvSchemes')	18
2.3.3/	Finite Volume Solution Schemes ('fvSolution').....	19
2.3.4/	Solution under-relaxation (Relaxation Factors)	20
2.3.5/	Initial and Boundary Conditions	20
3 /	<i>Numerical Sensitivity Studies</i>	22
3.1/	Reference Case Descriptions	22
3.1.1/	Grid Generation	23
3.1.2/	Key Numerical Parameters.....	24
4 /	<i>Benchmark Results - Resistance Prediction in Calm Water Condition</i>	25
4.1/	KCS Container Ship in Fixed Dynamic Condition	25
4.2/	DTMB 5415 Combatant in Fixed Dynamic Condition	28
4.3/	DTMB 5415 Combatant in Free Condition	34
4.3.1/	Resistance and Resistance Coefficients	34
4.3.2/	Sinkage and Trim.....	35
5 /	<i>Industrialisation of foamStar – Script Development & Validation</i>	38
6 /	<i>Conclusion</i>	38
7 /	<i>Future Work</i>	39
8 /	<i>References</i>	40

LIST OF THE FIGURES

Figure 2-1 Algorithm of foamStar (Monroy, Charles; Seng, Sopheak; Malenica, Sime, 2016).....	13
Figure 2-2 PIMPLE Algorithm in foamStar (LI, 2018).....	16
Figure 2-3 Overview of OpenFOAM Structure (Accessed from https://cfd.direct/openfoam/user-guide/v5-introduction/ on March 29, 2021).....	17
Figure 2-4 Typical Case Structure in OpenFOAM.....	17
Figure 2-5 Boundary Conditions of Flow Domain in foamStar	21
Figure 3-1 DTMB 5415 Model	22
Figure 3-2 Flow Domain of DTMB 5415 Combatant Hull	23
Figure 3-3 DTMB Mesh Views on Centreline Plane (a) profile view, (b) zoom on bow region and (c) zoom on stern region.....	23
Figure 3-4 DTMB Mesh on Transverse Sections (a) $x/L_{pp} = 0.1$ [-] and (b) $x/L_{pp} = 0.75$ [-].....	24
Figure 4-1 KCS Hull Model.....	25
Figure 4-2 Mesh on KCS Hull Model (a) Overall, (b) Stern Region and (c) Bow Region	26
Figure 4-3 Mesh View on Centerline Plane (a) Overall, (b) Stern Region and (c) Bow Region	27
Figure 4-4 Convergence History of Forces (KCS Hull)	27
Figure 4-5 Comparison of Wave Pattern (KCS Hull).....	28
Figure 4-6 Convergence History of Forces in DTMB Fixed Condition	29
Figure 4-7 Normalised Free Surface Elevation: Comparison with Experimental Results	30
Figure 4-8 Longitudinal Cut Locations in Far Field from Hull	30
Figure 4-9 Wave Profiles at (a) $y/L_{pp} = 0.082$ [-], (b) $y/L_{pp} = 0.172$ [-] and (c) $y/L_{pp} = 0.301$ [-].....	31
Figure 4-10 Velocity Field and Velocity Contour at Different Locations along Hull.....	32
Figure 4-11 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.2$ [-] (a) foamStar and (b) Experiment.....	32
Figure 4-12 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.6$ [-] (a) foamStar and (b) Experiment ...	33
Figure 4-13 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.8$ [-] (a) foamStar and (b) Experiment.....	33
Figure 4-14 Streamline and Cross-flow Vectors at $x/L_{pp} = 1.1$ [-] (a) foamStar and (b) Experiment ...	33
Figure 4-15 Comparison of Total Force in DTMB Free Condition.....	34
Figure 4-16 Comparison of Total Resistance Coefficient in DTMB Free Condition	35
Figure 4-17 Comparison of Non-dimensionalised Sinkage Results in DTMB Free Condition	36
Figure 4-18 Comparison of Trim Results in DTMB Free Condition.....	37



LIST OF THE TABLES

Table 2-1 Mesh Quality Criteria by 'checkMesh' in OpenFOAM.....	18
Table 2-2 Summary of Finite Volume Discretization Schemes "fvSchemes".....	19
Table 2-3 Summary of Finite Volume Solution Schemes 'fvSolution'.....	20
Table 2-4 Boundary Conditions used in foamStar/OpenFOAM.....	21
Table 2-5 Symbols to Boundary Conditions.....	21
Table 3-1 DTMB 5415 – Reference Case Description.....	23
Table 3-2 Grid Quality of Reference Case.....	24
Table 3-3 Water and Air Properties.....	24
Table 3-4 Numerical Parameters in foamStar.....	24
Table 4-1 KCS Container Model Parameters.....	26
Table 4-2 Comparison of Resistance Coefficients (KCS Hull).....	28
Table 4-3 DTMB Fixed Condition Settings.....	28
Table 4-4 Comparison of Resistance Coefficients in DTMB Fixed Condition.....	29
Table 4-5 DTMB in Free Condition Settings.....	34
Table 4-6 Total Force of foamStar and Experiment in DTMB Free Condition.....	34
Table 4-7 Resistance Coefficients and Percent Differences in DTMB Free Condition.....	35
Table 4-8 Sinkage and Trim Results and Comparison in DTMB Free Condition.....	36

1 / Introduction

In predicting the resistance of ship hulls, experiments in towing tanks have been mostly performed before the extensive use of CFD applications. Time and cost turn out to be key factors in consideration along with accuracy of results. High costs of experiments and limitations in testing several scenarios in towing tests led to an increased use of CFD codes for ship resistance and propulsion applications.

In Bureau Veritas Solutions (BVS) Marine and Offshore (M & O), hydrodynamic computations are being performed on practical industrial applications such as saving fuel consumption, energy efficiency and improving hull performance by optimising shapes. Hence, BVS has interests in developing in-house CFD codes in order to reduce license costs and to enable the engineers to contribute to the code development. One of these CFD tools is being jointly developed by Bureau Veritas (BV) and École Centrale de Nantes (ECN). It is based on OpenFOAM open-source libraries. Its name is *foamStar*.

1.1/ *foamStar* Description

foamStar is a CFD tool based on OpenFOAM open-source libraries for obvious reasons of accessibility and for its capability to simulate two-phase ship hydrodynamics flow problems. The nature of flow physics in ship hydrodynamics is a two-phase (water and air) incompressible and immiscible flow. The free surface between two phases can be monitored by interface tracking and capturing techniques. OpenFOAM's standard multi-phase solvers are implemented with volume of fluid (VOF) capturing method.

In addition, *foamStar* is integrated with Spectral Wave Explicit Navier-Stokes Equations (SWENSE) and Higher Order Spectral Method (HOS) in wave modelling processes. Thus, it has the capabilities of predicting ship hydrodynamic performances not just in calm water condition also in wave condition with high fidelity. Since OpenFOAM's standard solver "interDyMFoam" is implemented as background solver, dynamic behaviours of the ship can be captured well. In the present work this solver is used to couple the ship motions to the fluid solver.

Moreover, its environment is compatible with easy setup of simulation cases thanks to preset scripts, use of OpenFOAM's standard meshers or third party meshers, and post-processing in open-source visualisation tools such as paraView. These capabilities make it an interesting alternative to commercial software packages such as ISIS-CFD and STAR-CCM+.

1.2/ Purpose of Internship

Since industrial applications require accuracy and trustworthiness, the capabilities of the tool need to be verified and validated by conducting benchmarking campaigns where series of test computations are performed and results are analysed.

The purpose of the internship is to verify that the CFD tool called "*foamStar*/OpenFOAM" can provide trustworthy results, repeatable results and cost effective results in view of industrial use. This will be completed by performing series of computations on well-known benchmark test cases of ITTC and the industrial cases of Bureau Veritas Solutions (BVS) itself.

The three main objectives set as roadmap for this thesis are:

- ▶ Develop a good knowledge based on *foamStar* and its specificities
- ▶ Validation of *foamStar* use for industrial shipping applications through multiple benchmarks
- ▶ Develop user-friendly scripts for *foamStar* to perform different types of calculations

The work will be focused on the following key topics:

- ▶ Resistance in calm water condition
- ▶ Automation of calculation setup and launch
- ▶ Optimisation of hull forms



The thesis report first presents a literature review with detailed description of *foamStar* and OpenFOAM (philosophy, technical functionalities, and specificities). After that, the definition and selection of cases for studying numerical sensitivity and for conducting benchmarks are presented. Based on the results of the calculations, the influence of numerical parameters such as mesh renumbering, wave refinement zone shapes, number of PISO loops, relaxation factors and CPU decomposition are studied and the impact of these parameters on the accuracy of results are discussed. The most adapted settings will be recommended for future industrial applications. It is followed by the development of an automation script for ease of calculation setup and launch which will then be used to run a BVS optimisation case. Finally, conclusion and future work sections provide key outcomes of the work carried out during the internship.

2 / Literature Review

2.1/ General Introduction on OpenFOAM/foamStar

2.1.1/ OpenFOAM Description

OpenFOAM is distributed by the OpenFOAM Foundation, an organisation holding the copyright of the software and its documentation. The purpose of the Foundation is to manage and distribute OpenFOAM as free, open source software for the benefits of its users. It is a registered company, limited by guarantee based in England. OpenFOAM® is a registered trademark of OpenCFD Ltd, licensed to the OpenFOAM Foundation in 2011.

Open Source Field Operation and Manipulation (OpenFOAM®) is a framework for developing executable applications based on C++. It consists in 250 pre-built applications which can be divided into *solvers* and *utilities*. Solvers are designated to solve a specific problem in fluid (continuum) mechanics. Utilities are intended to perform tasks that involve data manipulation.

Solvers cover a wide range of problems in fluid dynamics and this coverage can be extended and customized by use and knowledge of underlying methods, physics and programming techniques (GREENSHIELDS, 2017). It creates a versatile and well suited foundation for CFD software development.

2.1.2/ Standard and Extended Solvers for Marine Applications

In OpenFOAM, there are several built-in standard solvers for various flow problems (GREENSHIELDS, 2017). In marine hydrodynamic applications, two-phase incompressible flow phenomena are particularly seen and the following solvers are the main references. The selection of solver depends on the consideration of physics to be modelled and the motion of the body of interest.

'interFoam' and 'interDyMFoam' Solvers

'interFoam' is a standard multiphase flow solver for two-phase incompressible, immiscible flow with the volume of fluid (VOF) method to capture the interface. Laminar or turbulent flow can be specified and the corresponding turbulence models can be utilized. There is no degree of freedom (DOF) considered nor solved. The model is assumed as fixed and no sinkage (heave) and no trim (pitch) results will be calculated.

'interDyMFoam' is also a standard multiphase flow solver for two-phase incompressible, immiscible flow using the volume of fluid (VOF) method. The main difference with the previous one is the fact that it includes parameters for mesh motion and mesh topology changes, even adaptive re-meshing can be performed. The dynamic effects can be considered in the simulation. The model is assumed as free and sinkage and trim can be captured well.

The resistance computations on KCS and container ship in calm water condition with two different turbulence models (k- ω SST and standard k- ω) are presented in (Shahzad, 2017). The solver 'interFoam' is used from 0 to 5 [secs] then switched to 'interDyMFoam' from 5 to 29 [secs] to capture the dynamic behaviour of ship. The high computational demands of wave generation by 'interFoam' is proven by grid study.

According to AIP Conference Proceedings 1919, 020038 (2017), the calm water resistance of JBC hull is estimated by using OpenFOAM solvers and compared with the experimental results. OpenFOAM's solvers 'LTSinterFoam', 'interFoam' and 'interDyMFoam' are applied separately in each simulation. It is concluded that 'interDyMFoam' solver significantly over predicts the sinkage motion, and resistance is over-estimated. In case of 'LTSinterFoam' and 'interFoam', heave and pitch motions are totally ignored and ship is in fixed position throughout the simulation. Results by 'interDyMFoam' solver are comparable with another RANS solver called SHIP_Motion solver.

According to (Liu, 2019), 'interFoam' solver is used for static drift simulations and 'interDyMFoam' solver is used for dynamic PMM (pure sway, pure yaw, combined yaw and drift) simulations.

‘waveFoam’ and ‘waveDyMFoam’ Solvers

The only difference with ‘inter(DyM)Foam’ solvers is the introduction of relaxation zones for wave calculations. ‘waveDyMFoam’ solver is based on ‘waveFoam’ solver with the addition of 6-DOF loop. In wave propagation with and without the presence of an object, ‘waveFoam’ solver is used in cases with static mesh and ‘waveDyMFoam’ for cases when mesh motion is required (Bruinsma, 2016).

‘navalFoam’ Solver (Naval Hydro Pack)

‘navalFoam’ under Naval Hydro Pack is a transient solver based on PIMPLE-algorithm. It is used to compute resistance coefficients, sinkage, trim and wave patterns of Japan Bulk Carrier (JBC) hull and MARIN systematic series fast displacement ship (FDS-5). The results and their comparison with published data can be found in (Tober, 2020).

In (Vukcevic & Jasak, 2014), accelerated steady state solver ‘steadyNavalFoam’ and transient solver ‘navalFoam’ are used for calm water resistance on US Navy Combatant DTMB 5415 and on KRISO Container Ship (KCS) for calm water resistance, forced sway motion and sea-keeping performance in head waves.

Other Solvers of Interest

The flow around a marine propeller can be investigated by means of ‘simpleFoam’ and ‘pimpleDyMFoam’. The solver ‘simpleFoam’ is a steady-state solver in which the flow is solved in a relative coordinate system that follows the blade movement according to the Moving Reference Frame (MRF) method. The original Navier-Stokes equations are modified. The solver ‘pimpleDyMFoam’ is a transient code which solves the time-accurate Navier-Stokes equations through an iteration procedure within each time step by enabling the usage of under-relaxation in the transient computation (improve stability and allow larger time step) (Turunen, Siikonen, Lundberg, & Bensow, 2014).

The optimization of the design of accelerating and decelerating nozzles for ducted propeller applications is performed by using the efficient RANS solver ‘simpleFoam’ of OpenFOAM and modeFrontier optimization environment in (Gaggero, Villa, Tani, Viviani, & Bertetta, 2017).

For propeller in oblique flow inside tunnel in wetted and cavitating conditions, the modified ‘interPhaseChangeDyMFoam’ solver is utilised in (Asnaghi, Feymark, & Bensow, 2015).

In (Jasak, Hrvoje; Tukovic, Zeljko, 2010), it says the problems of manoeuvring (incompressible flow), aircraft (compressible transonic flow) or a floating object (volume-of-fluid free surface flow) will have the solid body motion which is unknown and part of solution. When the object moves as solid body, the flow domain around it does not and the prescribed surface motion needs to be accommodated by mesh deformation. It can be defined by ‘sixDofMotion’ dynamic class in OpenFOAM.

2.1.3/ foamStar and Its Specifications

As illustrated above, OpenFOAM provides a wide range of uses for CFD computations and marine applications. However, in order for ECN and BV to get their own version exclusively focused on marine applications *foamStar* development started about ten years ago. The philosophy and goal behind developing *foamStar* can be stated as:

- ▶ to fulfil the increasing needs for multi-approach wave modelling in ship resistance and propulsion,
- ▶ to perform the computations with reasonable cost in aspects of licenses and computation (CPU) time,
- ▶ to obtain accuracy, reliability and trustworthiness in results for industrial marine applications.

foamStar is a solver for ship resistance predictions in calm water and wave condition. It can also be used to evaluate the open water performance of marine propellers. This solver is based on

'interDyMFoam' solver of OpenFOAM-5x standard multiphase flow solver for two incompressible immiscible fluids. *foamStar* itself is comprised of several modules and solver types developed throughout the years by ECN's PhD students, BV's doctors and research engineers. The implemented solvers/modules aim to meet the expectations of both the academic research and industrial necessities. Some of those are work in progress and validations on naval applications are compulsory.

In addition, scripts are being developed to facilitate the setup of standard marine CFD computations and save time.

Discussion on wave generation methods

In *foamStar*, the additional packages of higher order spectral method (HOS, SWENSE) are being implemented for wave generation. The wrapper program called 'Grid2Grid' is also being coupled in view of generating nonlinear wave field by Inverse Fourier Transform and by quick spline module.

In literature, difficulties linked to wave modelling in numerical wave tanks (NWT) can be overcome in two ways:

- ▶ Developing generation-absorption boundary conditions which aim at generating incident waves and absorbing the diffracted waves at the same time or,
- ▶ Treating wave generation and absorption separately as stated in (Li, et al., 2019).

The second approach is implemented as relaxation zone method in (Monroy, Charles; Seng, Sopheak; Malenica, Sime, 2016) and as SWENSE method in (LI, 2018).

According to this paper, the modifications in OpenFOAM's standard 'interDyMFoam' solver can be seen in the relaxation zone, wave models and fluid-structure interaction. The solution algorithms for time and non-linear iterations are described as follows:

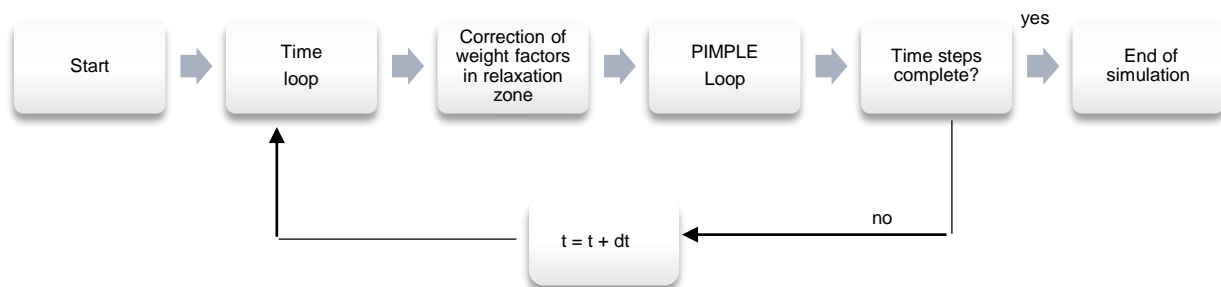


Figure 2-1 Algorithm of *foamStar* (Monroy, Charles; Seng, Sopheak; Malenica, Sime, 2016)

In (Zhaobin Li, et al., 2019) , the SWENSE method is used for wave structure interaction problems to obtain efficient incident wave solution and coupled with viscous solver.

The higher order spectral (HOS) method is also used for the study of wave propagation in open domains as it models the non-linearity of free surface up to wave breaking limit. There are two open-source solvers developed in LHEEA Lab namely HOS-Ocean and HOS-NWT.

2.2/ Technical Review of Key OpenFOAM/foamStar Aspects

2.2.1/ Mathematical Introduction to Finite Volume Method in CFD

Let's consider that the infinitesimal fluid element has density ' ρ ' and three velocity components ' V (u, v, w)' respectively in (x, y, z) direction. In hydrodynamics applications, the fluid can be assumed to be incompressible. The continuity equation for incompressible fluid becomes

$$\frac{\partial V_k}{\partial x_k} = 0 \quad \text{Eq. 1}$$

The momentum equation which includes inertia terms, pressure terms and viscosity terms in three-dimensions can be described as follows (Le Touze, 2020):

$$\frac{\partial V_i}{\partial t} + \frac{\partial V_i}{\partial x_k} V_k = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\mu}{\rho} \left(\frac{\partial^2 V_i}{\partial x_k \partial x_k} \right) + f_i \quad \text{Eq. 2}$$

Where f_i represents external forces. These sets of equations are known as Navier-Stokes (NS) equations. The equations can be numerically solved by Reynolds-Averaging the NS equations (RANS) approach because not all turbulence details are of interest for engineering applications. After Reynolds averaging of Eq. 1 and Eq. 2, RANS equation sets can be described as follows:

$$\frac{\partial \bar{V}_k}{\partial x_k} = 0 \quad \text{Eq. 3}$$

$$\frac{\partial \bar{V}_i}{\partial t} + \frac{\partial \bar{V}_i}{\partial x_k} \bar{V}_k = -\frac{1}{\rho} \frac{\partial}{\partial x_i} \left(\bar{P} + \frac{2}{3} \rho k \right) - \frac{\partial \overline{V_i V_k}}{\partial x_k} + \frac{\mu_{eff}}{\rho} \left(\frac{\partial^2 \bar{V}_i}{\partial x_k \partial x_k} \right) + f_i \quad \text{Eq. 4}$$

Where μ_{eff} is the effective dynamic viscosity ($\mu_{eff} = \mu + \rho \nu_t$), μ the fluid dynamic viscosity, ν_t the turbulent viscosity, k the turbulent viscosity. The turbulent viscosity and the kinetic viscosity can be evaluated by using different models which are presented in the following section.

2.2.2/ Turbulence Models

The flow phenomena in naval applications can be hypothesized as laminar/ turbulent, incompressible flow with single phase or multi-phase. For turbulent flows, depending on the level of turbulence scale, simulation types can be divided as Reynold's Averaging of Navier-Stokes (RANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). Majority of naval CFD applications are implemented with RANS approach as it provides a good compromise between the accuracy and the computation cost.

In RANS approach, turbulence is modelled by eddy viscosity models. There are widely used turbulence models such as $k-\epsilon$, $k-\omega$ SST and Spalart-Allmaras.

In (Le Touze, 2020), there are several eddy viscosity models available and can be classified as:

- ▶ zero-equation/algebraic models (e.g. mixing length, Baldwin-Lomax)
- ▶ one-equation models (e.g. Spalart-Allmaras, k -model, Baldwin-Lomax, Wolfstein)
- ▶ two-equation models (e.g. $k-\epsilon$, $k-\omega$, $k-\omega$ SST)
- ▶ three-equation models (e.g. $k-\epsilon-A$)
- ▶ four-equation models (e.g. v^2-f model)

The two-equation model $k-\omega$ SST is mainly used in *foamStar* computations. It is a two equation model defining the turbulence kinetic energy, k , and turbulence specific dissipation rate, ω for incompressible flows. The aim is to overcome the deficiencies of the standard k - ω model with respect to dependency on the freestream values of k and ω using $k-\epsilon$ to solve the far field while keeping a good capture of flow separation next to the bodies.

2.2.3/ Finite Volume Discretisation

In order to solve the governing equations numerically, the continuum fluid domain is divided into small discrete volumes known as spatial discretisation and these governing equations are applied on these volumes so that the flow variables can be solved over the discrete points (cell centre) known as equation discretisation.

In spatial discretisation, the domain can be discretised by using different cell shapes such as prisms, tetrahedrons, hexahedrons and polyhedrons. Since the geometries involved have complex shapes in nature, unstructured grids are mainly generated in marine hydrodynamic applications.

In spatial and temporal equation discretisation, the governing equations are transformed into matrix systems of linear algebraic equations by rewriting over small volumes. The discretisation schemes for different terms especially convection and diffusion terms in RANS equations need to be carefully selected depending on the stability and accuracy of the schemes. These schemes can be said implicit or explicit by the dominance of the diagonal terms in the matrix. The temporal schemes can be said explicit or implicit. It results into diagonal matrix resolution for the first ones and matrix resolution with off-diagonal terms for the seconds.

Along with the discretization process, the resolution domain is bounded. The resulting boundaries lead to some specific terms in the matrix systems. The temporal discretisation is bounded imposing initial fields at the beginning of a simulation and the spatial discretisation is bounded with imposed terms in the matrix systems. These terms are defined using the so called "boundary conditions". The finite volume spatial and temporal discretisation schemes and boundary conditions available in *foamStar*/OpenFOAM for different terms in flow equations will be presented in the subsequent sections.

2.2.4/ Linear Solver Algorithms

In RANS equations, the pressure and velocity terms are coupled. Consequently, different strategies exist in order to build the discrete matrix systems of linear equations.

The velocity-pressure decoupling algorithms known as Pressure-Implicit Split-Operator (PISO), Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) or combined PISO and SIMPLE (PIMPLE) are required to implement: the velocity field is predicted by solving momentum equations, the pressure field is solved using the momentum equation, the continuity equations and the predicted velocity. PISO and PIMPLE algorithms are for transient problems. SIMPLE algorithm is for steady-state problems.

PIMPLE algorithm is implemented in *foamStar* and the algorithm is described in Figure 2-2. There are two loops in PIMPLE algorithm: the inner loop solves the velocity-pressure coupling with PISO algorithm and the outer loop uses SIMPLE algorithm to deal with the non-linear dependency between the velocity field and free surface position (LI, 2018).

With such algorithm, several matrix resolutions occur. To solve the matrices (both large and sparse), there are several methods available namely Gauss method, Jacobi method, Gauss-Seidel method, relaxation methods, Gradient Conjugate method, Bi-Conjugate Gradient Conjugate method, Generalised Minimal Residual Method (GMRES) and multigrid methods (Gentaz, 2020).

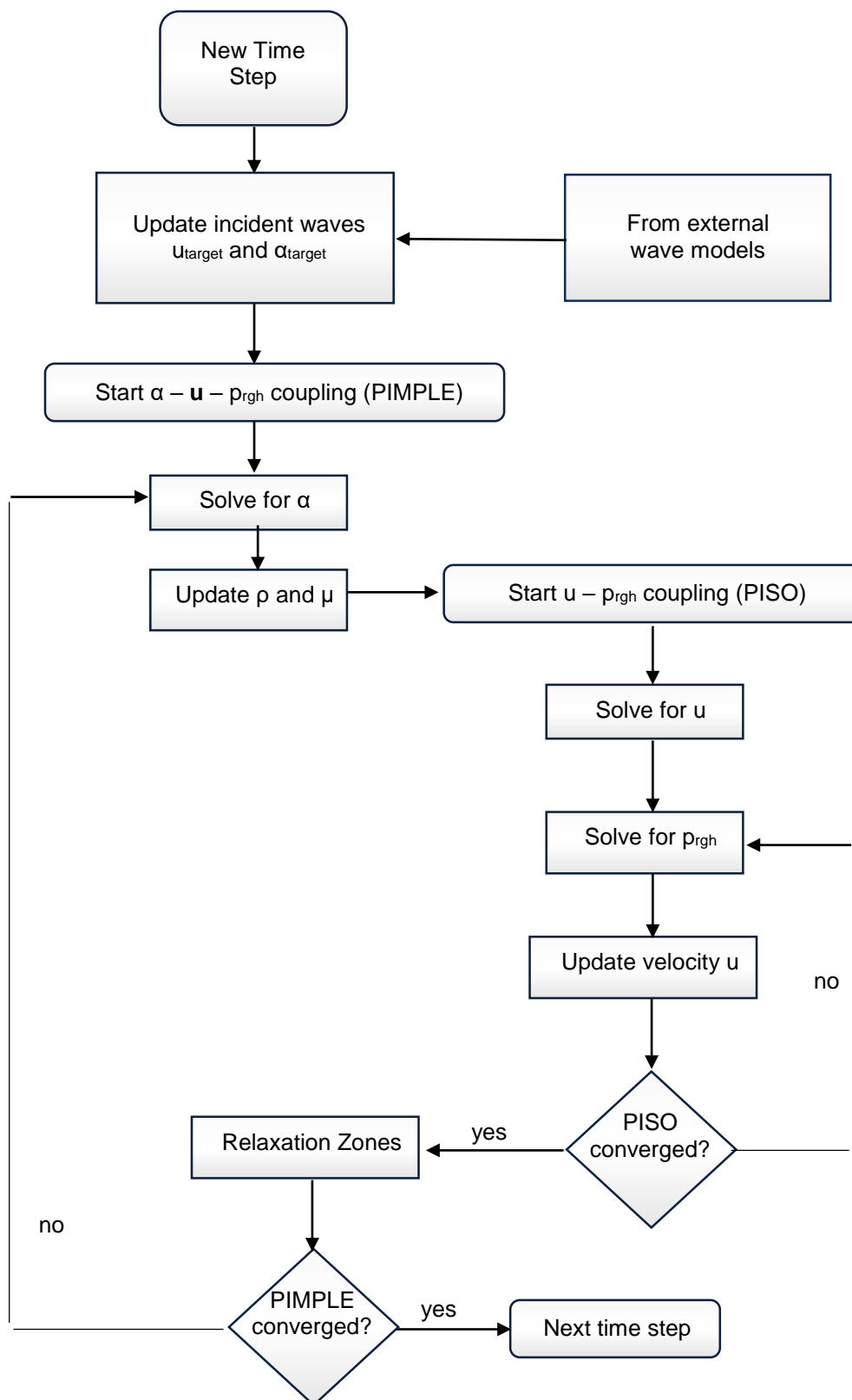


Figure 2-2 PIMPLE Algorithm in foamStar (LI, 2018)

2.3/ Basic Case Architecture in OpenFOAM – Focus on Utilities

The figure hereafter presents the structure of the OpenFOAM software divided in the three key steps of computations.

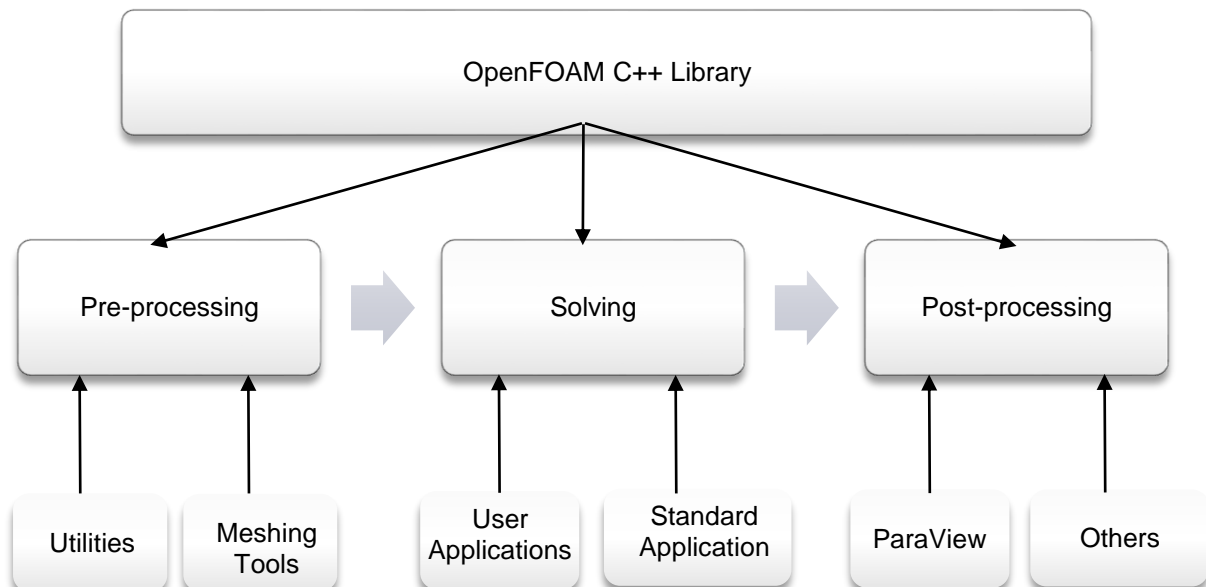


Figure 2-3 Overview of OpenFOAM Structure (Accessed from <https://cfd.direct/openfoam/user-guide/v5-introduction/> on March 29, 2021)

Whenever starting a case in OpenFOAM, there must be three folders called 0/, constant/ and system/. The corresponding dictionaries are required to place under these folders in prior the calculation. The structure is shown in Figure 2-4.

The three folders (in blue) and the corresponding dictionaries (in green) are as follows:

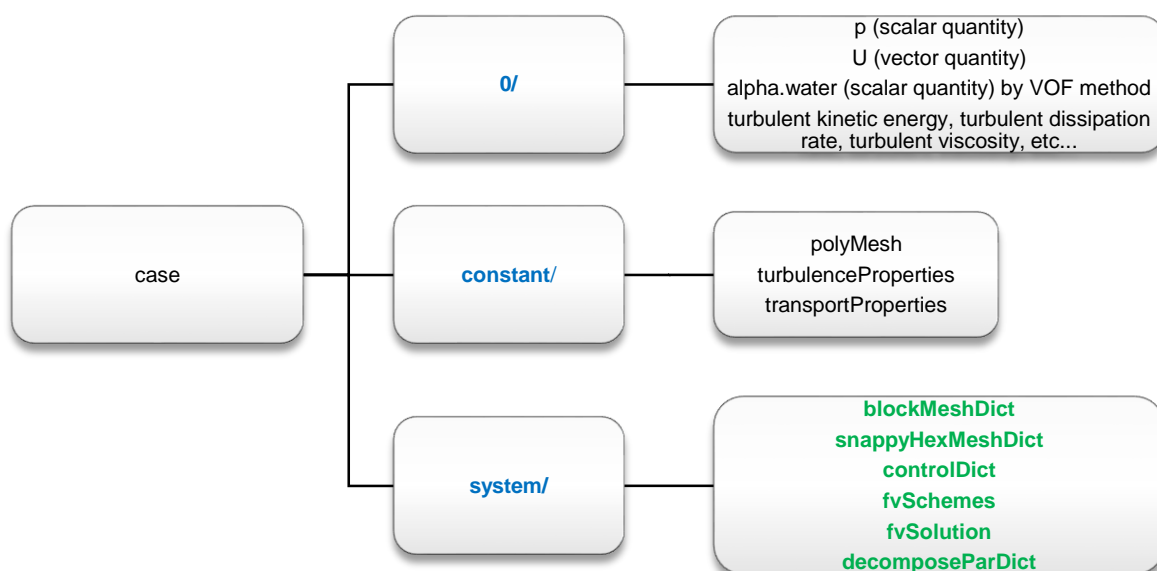


Figure 2-4 Typical Case Structure in OpenFOAM

These dictionaries contain various information of:

- ▶ initial and boundary conditions
- ▶ transport properties of water/air
- ▶ mesh data and mesh quality
- ▶ finite volume discretization schemes
- ▶ solution algorithms and time step parameters

2.3.1/ Mesh Generation Utilities

There are two generic meshers in OpenFOAM mainly used in applications, namely:

- ▶ blockMesh
- ▶ snappyHexMesh

The principle behind ‘blockMesh’ is to decompose the domain geometry into a set of one or more three dimensional, hexahedral blocks. Edges of the blocks can be straight lines, arcs or splines. The local coordinated system of the block can be established by right-hand rule. With this meshing utility, single-block or multi-block hexahedral mesh can be generated (GREENSHIELDS, 2017).

The tool ‘snappyHexMesh’ is a mesh generator for complex geometry (e.g. mesh around vehicles). It is a mesh generation utility which generates three-dimensional meshes including *hexahedra (hex)* & *split-hexahedra (split-hex)*. The mesh can automatically be generated from triangulated surface geometries or tri-surfaces in Stereolithography (STL) and Wavefront Object (OBJ) (GREENFIELDS, 2017).

The process steps of ‘snappyHexMesh’ can be summarised as follows:

- 1) Generation of background hex mesh by ‘blockMesh’ utility
- 2) Cells splitting by feature edges and surfaces
- 3) Removal of unnecessary cells inside domain
- 4) Cells splitting in specified regions
- 5) Snapping to surfaces
- 6) Insertion of cell layers
- 7) Control of mesh quality

At the end of the mesh generation, the command ‘checkMesh’ can be used to verify its validity and show the grid quality. The characteristics of cells such as aspect ratio, non-orthogonality and skewness are compared with standard OpenFOAM criteria (GREENFIELDS, 2017), listed in Table 2-1.

Mesh Characteristics	Criteria
Max. Cell Non-orthogonality [deg]	70
Max. Cell Skewness (Internal)	4
Max. Cell Skewness (Boundary)	20

Table 2-1 Mesh Quality Criteria by ‘checkMesh’ in OpenFOAM

2.3.2/ Finite Volume Discretization Schemes (‘fvSchemes’)

In order to discretise the governing equations by finite volume method in OpenFOAM, the proper definition and selection of schemes for each term in the governing equations need to be specified in ‘fvSchemes’ dictionary. This dictionary provides wide selections of discretisation schemes to users, the key ones can be summarised as follows:

Schemes	Keywords	Maths Symbols	Available Schemes
Time discretization scheme	'ddtSchemes'	$\frac{\partial}{\partial t}$	<ul style="list-style-type: none"> • steadyState • Euler • backward • crankNicolson • localEuler
	'd2dt2Schemes'	$\frac{\partial^2}{\partial t^2}$	<ul style="list-style-type: none"> • Euler
Gradient discretization scheme	'gradSchemes'	∇	<ul style="list-style-type: none"> • Gauss linear • cellLimited Gauss linear 1 (to improve boundedness & stability) • leastSquares • Gauss cubic
Convective terms discretization scheme	'divSchemes'	$\nabla \cdot$	<ul style="list-style-type: none"> • linear • linearUpwind • LUST (75% linear, 25% linearUpwind) • limitedLinear:linear • upwind
Laplacian discretization (second derivatives in space)	'laplacianSchemes'	∇^2	<ul style="list-style-type: none"> • Gauss scheme
Interpolation schemes	'interpolationSchemes'		<ul style="list-style-type: none"> • Linear interpolation
Surface normal gradient discretisation	'snGradSchemes'		

Table 2-2 Summary of Finite Volume Discretization Schemes "fvSchemes"

2.3.3/ Finite Volume Solution Schemes ('fvSolution')

The main dictionary is denoted as 'fvSolution' and there are also sub-dictionaries for solvers, relaxation factors, and the parameters for PISO, SIMPLE or PIMPLE algorithms. Matrices are sparse in nature and solvers use iterative procedures to solve them. It is necessary to define stopping criteria for these iterative procedures. There are three solvers stopping criteria in OpenFOAM:

- ▶ Residual falls below solver tolerance, 'tolerance'
- ▶ Ratio of current to initial falls below solver relative tolerance, 'relTol'
- ▶ Number of iterations exceeds maximum number of iterations, 'maxIter'

Types of solvers implemented in OpenFOAM are summarised as follows:

Types of Solvers	Remarks
PCG/PBiCGStab PCG/PBiCG	Precondition of matrices in conjugate gradient solvers (preconditioner) <ul style="list-style-type: none"> • DIC/DILU • FDIC • diagonal • GAMG • none
smoothSolver	<ul style="list-style-type: none"> • GaussSeidel • symGaussSeidel • DIC/DILU • DICGaussSeidel 'GaussSeidel' and 'symGaussSeidel' are preferred.
GAMG	<ul style="list-style-type: none"> • first generate a quick solution on a mesh with a small number of cells • mapping this solution onto a finer mesh to obtain accurate solution
diagonal	-

Table 2-3 Summary of Finite Volume Solution Schemes 'fvSolution'

2.3.4/ Solution under-relaxation (Relaxation Factors)

The relaxation factors are used for improving stability of a computation. The factor is applied to stabilise calculations by limiting the rate of change of both fields and equations either by modifying the solution matrix and source prior to solving or by modifying the field directly.

This factor has the range: $0 < \alpha \leq 1$. When the factor 'α' is decreased, the under-relaxation increases. When α is equal to 1, the matrix is forced to become diagonally equal. The solution doesn't change when α is equal to 0. The field under-relaxation can be specified as:

$$Q^{n,used} = Q^{n-1} + \alpha (Q^{n,predicted} - Q^{n-1}) \quad \text{Eq. 5}$$

Low relaxation factors help for computation stability but convergence becomes slower. In contrast, high relaxation factors show computation potentially unstable but it converges faster.

2.3.5/ Initial and Boundary Conditions

Initial and boundary conditions are required in solving governing equations numerically. Initial conditions are defined under 0/ folder in *foamStar*. This folder includes initial values of pressure field, velocity field, alpha water by VOF method, and turbulent quantities such as turbulent kinetic energy, turbulent dissipation rate and turbulent viscosity, etc.

Boundary conditions are required to define prior to solving the flow equations. In naval CFD calculations, certain types of boundary conditions such as solid body (non-slip wall), velocity inlet, pressure outlet, symmetry plane and far field boundary conditions are assigned in order to represent the flow physics correctly. Since the flow around the hull is a two-phase flow (water and air), free surface boundary condition must also be satisfied.

The simulation can be performed in half or full domain. When the geometry is symmetric and the flow around the hull can be assumed symmetry, then the simulation can be performed in half domain. In this way, the computation time is also halved.

The typical boundary conditions used in *foamStar* calculations presented in this report are illustrated in Figure 2-5.

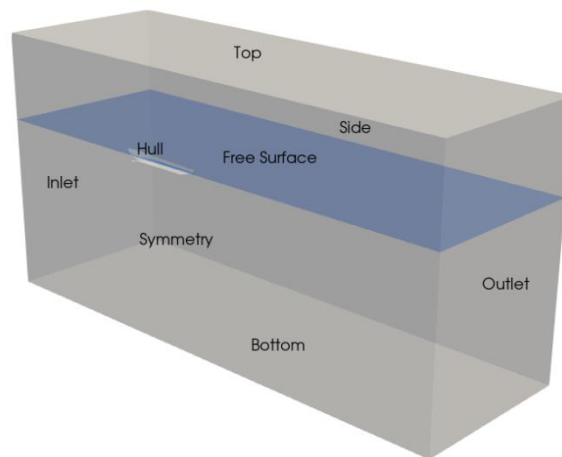


Figure 2-5 Boundary Conditions of Flow Domain in *foamStar*

The following boundary types to corresponding patches are used for calm water resistance calculations in *foamStar* and are tabulated as follows.

Boundary	U	p _{grh}	alpha.water	k	vt	ω
Inlet	WV	FFP	WA	FV	calculated	ZG
Outlet	WV	FFP	WA	FV	calculated	ZG
Side	WV	FFP	WA	FV	calculated	ZG
Symmetry	SP	SP	SP	SP	SP	SP
Top	PIOV	TP	IO	FV	calculated	ZG
Bottom	FV	FFP	ZG	FV	calculated	ZG
Solid Wall	MWFV	FFP	ZG	KQRWF	NUSWF	OWF

Table 2-4 Boundary Conditions used in *foamStar/OpenFOAM*

WV	waveVelocity	FFP	fixedFluxPressure
SP	symmetryPlane	TP	totalPressure
PIOV	pressureInletOutletVelocity	WA	waveAlpha
FV	fixedValue	IO	inletOutlet
MWFV	movingWallAndFrameVelocity	ZG	zeroGradient
KQRWF	kqRWallFunction	NUSWF	nutUSpaldingWallFunction
OWF	omegaWallFunction		

Table 2-5 Symbols to Boundary Conditions

Note for users:

When generating grid in Hexpress, the boundary and ship surfaces should be named in the following way. For the domain, the boundary surfaces are defined as: XMAX as inlet patch, XMIN as outlet patch, YMAX as side patch, YMIN as symmetry patch, ZMAX as top patch and ZMIN as bottom respectively. For hull surfaces, they should be named by starting with "SHIP_".

3 / Numerical Sensitivity Studies

In this chapter, series of computations will be performed in *foamStar* by varying grids and numerical parameters to figure out robustness and sensitivity. Since this is the first use of *foamStar* for BVS, the sensitivity studies and benchmark cases will be calm water resistance cases prior to any wave applications. The aim is to derive best practices in the model setup of resistance cases afterwards. The *foamStar* version used is '*foamStar Beta + forward speed*'.

The key topics covered in sensitivity studies on numerical setup are:

- ▶ Cell Renumbering
- ▶ Y+ Approach for the mesh
- ▶ Wave Refinement Zone near Free Surface
- ▶ Influence of PISO Iterations

General comments will be derived based on the results of sensitivity studies. All sensitivity cases are carried out on DTMB 5415 hull model because there are several results available in literature and especially R & D project 'Bassin Numerique' which involved with many industrial partners including BVS.

3.1/ Reference Case Descriptions

DTMB 5415 combatant model is used as the basic model for the sensitivity study. The experimental results are taken from (Olivieri, Pistani, Avanzini, F.Stern, & Penna, 2001). Table 3-1 shows the model parameters and the test condition. The following section presents the key information on the reference case used.



Figure 3-1 DTMB 5415 Model

Model Scale, λ	24.8	[-]
Length waterline, Lwl	5.727	[m]
Overall Length, LOA	6.172	[m]
Breadth, B	0.769	[m]
Draft, T	0.248	[m]
Displacement, Δ	0.555	[t]
Volume, V	0.554	[m ³]
Wetted Surface Area, S	4.858	[m ²]
Model speed, V_m	2.097	[m/sec]
Froude Number, Fn	0.28	[-]
Reynolds' Number, Re	1.195E+07	[-]

Test Condition	<ul style="list-style-type: none"> • Bare Hull • Calm Water • Still Water Position (free sinkage and trim) • Grid with High Y+
----------------	--

Table 3-1 DTMB 5415 – Reference Case Description

3.1.1/ Grid Generation

The grid is generated in Hexpress in accordance with the BVS industrial guidelines. The grid of the domain is shown in Figure 3-2. The overall flow domain has $6L_{pp} \times 2L_{pp} \times 3L_{pp}$. The local refinements are carefully defined on hull surfaces with high curvature (especially in bow and stern regions) to capture the flow phenomena accurately. Free surface and wave refinements are also added at the free surface level.

The mesh quality is checked in Hexpress to make sure there are no negative cells, concave cells and twisted cells. The maximum non-orthogonality in OpenFOAM can be checked in Hexpress. Afterwards, Hexpress mesh is converted into OpenFOAM format.

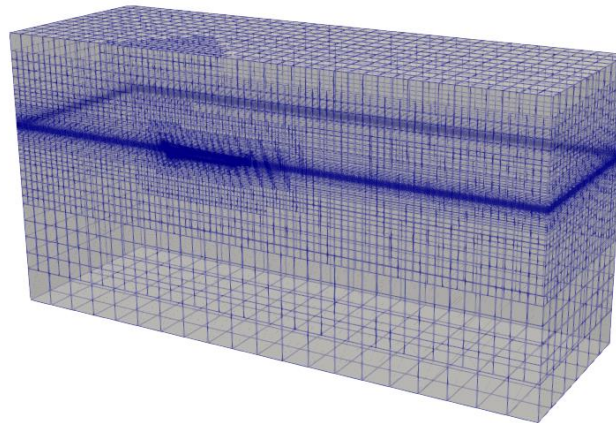


Figure 3-2 Flow Domain of DTMB 5415 Combatant Hull

The mesh on centreline planes and transverse sections are presented in Figure 3-3 and Figure 3-4.

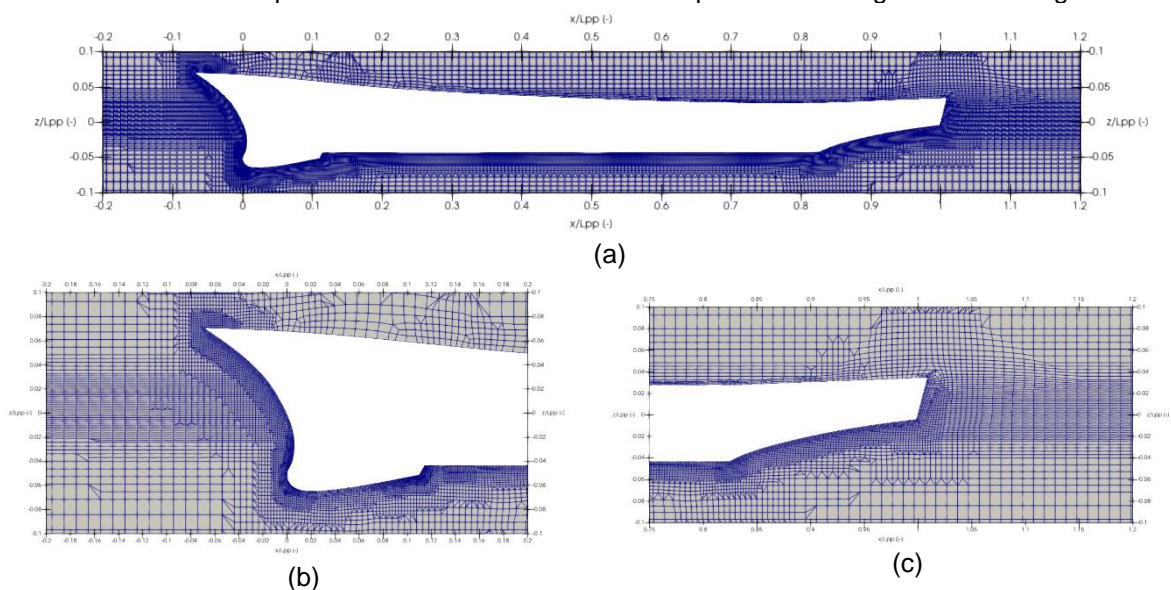


Figure 3-3 DTMB Mesh Views on Centreline Plane (a) profile view, (b) zoom on bow region and (c) zoom on stern region

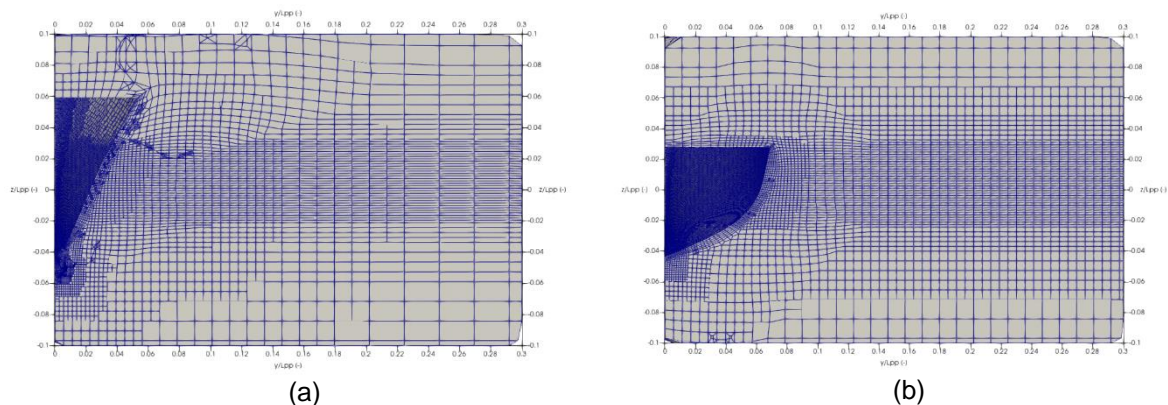


Figure 3-4 DTMB Mesh on Transverse Sections (a) $x/L_{pp} = 0.1$ [-] and (b) $x/L_{pp} = 0.75$ [-]

The grid quality is also analysed in *foamStar* by “checkMesh”. In Table 3-2, it can be seen that the grid has maximum non-orthogonality of 84 [deg] and the maximum skewness of 4.16. When comparing to OpenFOAM criteria, it is found that maximum non-orthogonality and skewness values are higher than OpenFOAM’s criteria: $84 > 70$ [deg] and $4.16 > 4$ respectively. *It is important to note that a good quality mesh in Hexpress will most likely not be sufficient in foamStar.*

Cells in millions	1.47
Max Aspect Ratio	57
Max Non-orthogonality [deg]	84
Avg Non-orthogonality [deg]	9.1
Max Skewness	4.16

Table 3-2 Grid Quality of Reference Case

3.1.2/ Key Numerical Parameters

The following numerical parameters have been applied to *foamStar* as default values. Water and air properties are shown in Table 3-3. The other numerical parameters such as time step, flow time, relaxation factor, no. of PIMPLE, PISO and non-orthogonal correction loops are adopted as tabulated in Table 3-4.

Properties	Water	Air
ρ [kg/m ³]	998.5	1.2
μ [Pa-sec]	1.003E-03	1.850E-05
ν [m ² /sec]	1.005E-06	1.542E-05

Table 3-3 Water and Air Properties

Parameter	Value	Unit
Time Step	0.01	[sec]
Flow Time	20	[sec]
No of Cores	12	[cores]
No. of PIMPLE Loops	8	[-]
No. of PISO Loops	3	[-]
No. of Non-orthogonal Correction	1	[-]
Relaxation Factor	0.9	[-]

Table 3-4 Numerical Parameters in *foamStar*

4 / Benchmark Results - Resistance Prediction in Calm Water Condition

KRISO container ship (KCS) and DTMB 5415 combatant hulls are selected to benchmark *foamStar* because these cases are stated by the International Towing Tank Committee (ITTC) as recommended benchmarks for CFD validation for resistance and propulsion (ITTC, 1996). Both KCS and DTMB hulls are well-known hull forms which have been extensively studied and are also selected test cases in Gothenburg 2000 workshop. These hull forms can be adopted as the good starting point with a lot of references shown in (ITTC, 1996).

KRISO container ship hull is a modern commercial hull form with the bulbous bow and stern bulb (Kim, Van, & Kim, 2001). The model has the length between perpendiculars of 7.2786 [m] corresponding to the scale ratio of $\lambda = 31.6$ [-].

DTMB 5415 has the transom stern and the bulbous bow of peculiar shape which is intended to accommodate the sonar (Olivieri, Pistani, Avanzini, F.Stern, & Penna, 2001). The model has the length between perpendiculars of 5.72 [m] corresponding to the scale ratio of $\lambda = 24.8$ [-].

There are three main areas of interest in selection of benchmarking cases to validate the capability of *foamStar*/OpenFOAM:

- ▶ resistance prediction in calm water condition
- ▶ optimisation of hull forms
- ▶ added resistance prediction in wave condition

The work presented in this section focuses on the first point which is a required validation step prior to any of the two others.

4.1/ KCS Container Ship in Fixed Dynamic Condition

The KCS bare hull with rudder in calm water condition at Froude 0.26 [-] is computed and analysed in fixed-dynamic position. The vessel is fixed at the given sinkage with zero trim, and without allowing to have sinkage and trim during the computation. The calculation is performed with 12 cores. The main information on the model and simulated case are presented in the table hereafter.



Figure 4-1 KCS Hull Model

Model Scale, λ	31.6	[-]
Length waterline, L_{wl}	7.3561	[m]
Length between perpendiculars, L_{pp}	7.2786	[m]
Breadth, B	1.0190	[m]
Draft, T	0.3418	[m]
Displacement, Δ	1.6488	[t]
Volume, V	1.6488	[m ³]
Wetted Surface Area, S	9.5528	[m ²]
Model speed, V_m	2.1970	[m/sec]
Froude Number, Fn	0.26	[-]
Reynolds' Number, Re	1.40E+07	[-]
Test Condition	<ul style="list-style-type: none"> • Bare Hull with Rudder • Calm Water • Fixed Dynamic Condition (sinkage imposed, zero trim) 	

Table 4-1 KCS Container Model Parameters

The grid generated is illustrated in Figure 4-2 and Figure 4-3.

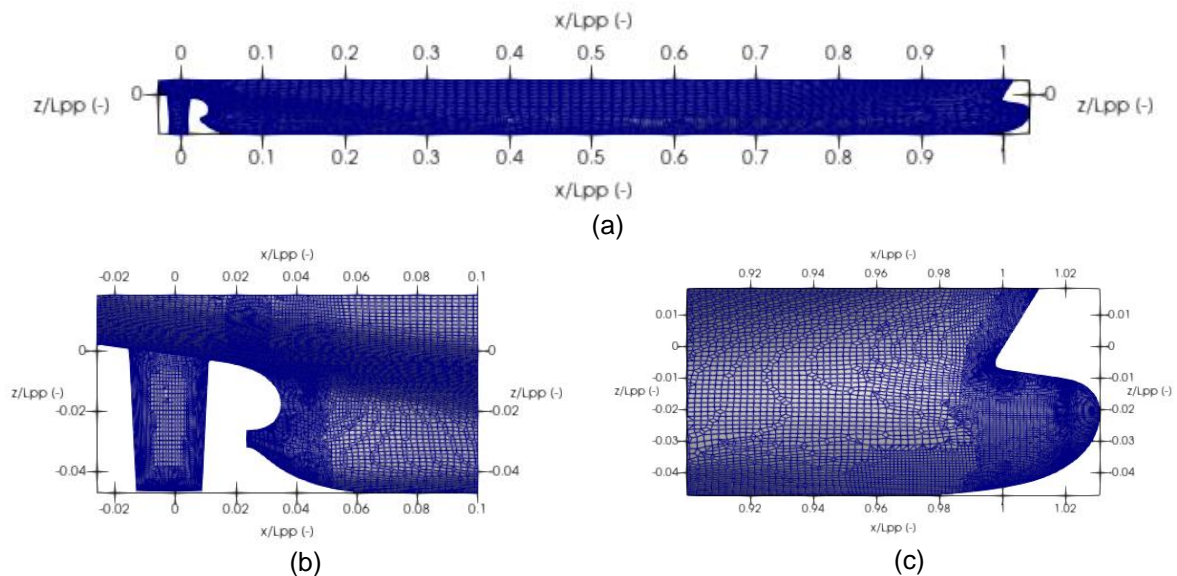


Figure 4-2 Mesh on KCS Hull Model (a) Overall, (b) Stern Region and (c) Bow Region

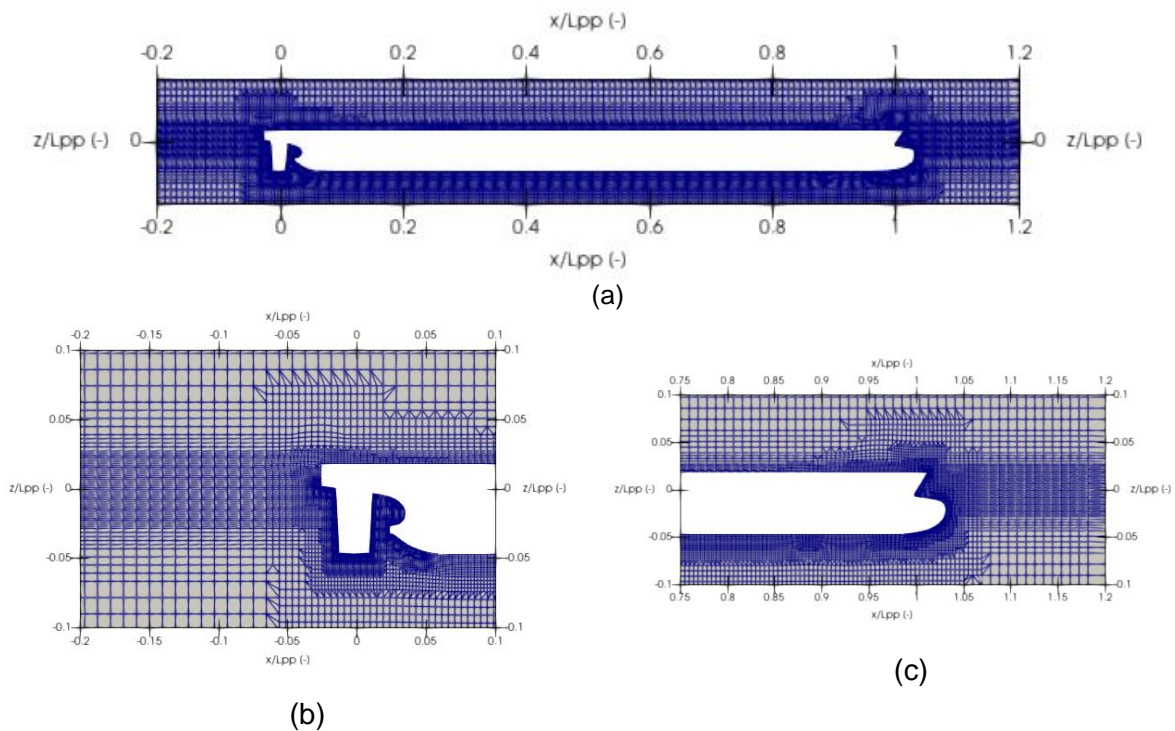


Figure 4-3 Mesh View on Centerline Plane (a) Overall, (b) Stern Region and (c) Bow Region

Presentation and discussion of the results

The viscous, pressure and total forces of half-body in the longitudinal direction are plotted in Figure 4-4. In viscous force, convergence is achieved after 1000 iterations (10 [sec]). In pressure force, there are some small oscillations remaining after 1600 iterations (16 [sec]).

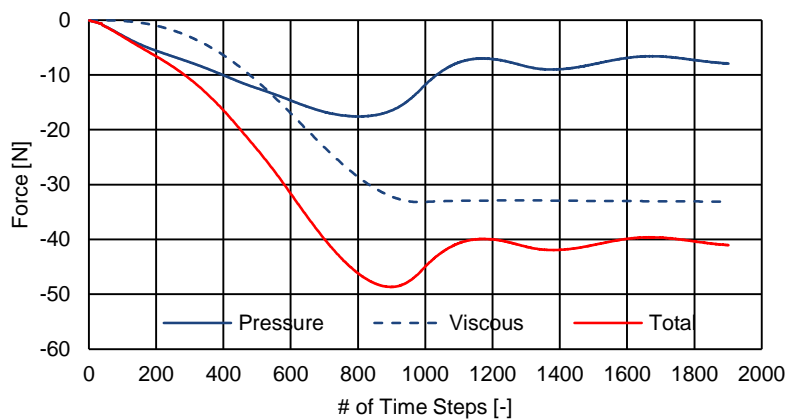


Figure 4-4 Convergence History of Forces (KCS Hull)

Experimental and computational results are compared and the differences are analysed as shown in Table 4-2. It is observed that there are 15.23 [%] difference in residuary resistance coefficient, 1.47 [%] in frictional resistance coefficient and overall 1.99 [%] in total resistance coefficient in comparison with experimental results.

	EFD	foamStar	Difference [%]
$C_f [-] \times 10^3$	2.83	2.872	1.47
$C_r [-] \times 10^3$	0.725	0.6145	-15.23
$C_t [-] \times 10^3$	3.557	3.486	-1.99

Table 4-2 Comparison of Resistance Coefficients (KCS Hull)

According to Table 4-2, it can be said that the total resistance in *foamStar* computation for KCS hull is well estimated within 2 [%] deviation. Free surface contours are investigated as illustrated in Figure 4-5. It is seen that the generated wave pattern well agrees with the experimental results.

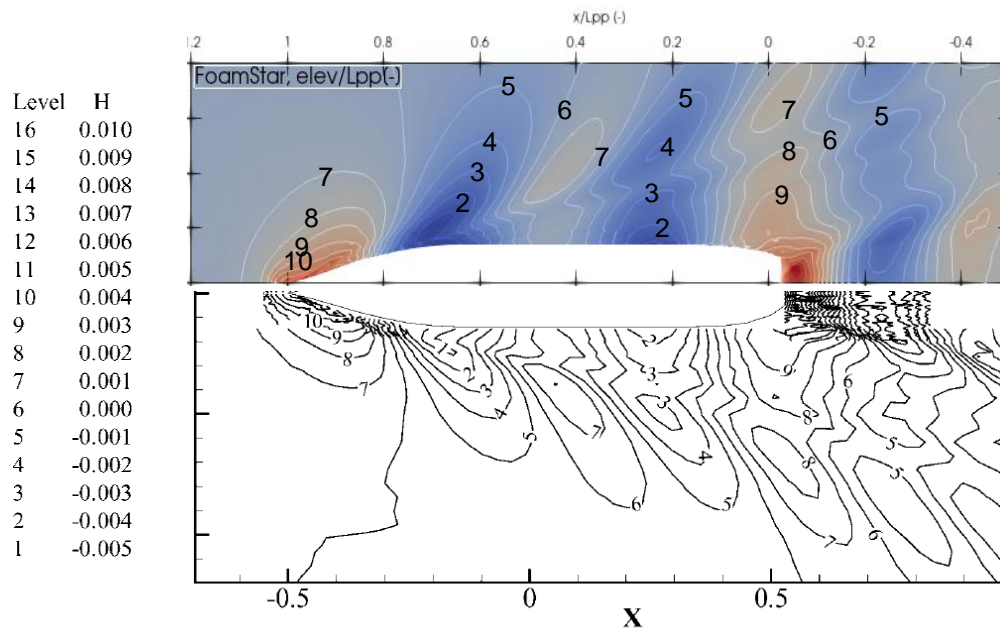


Figure 4-5 Comparison of Wave Pattern (KCS Hull)

4.2/ DTMB 5415 Combatant in Fixed Dynamic Condition

The DTMB 5415 bare hull in calm water condition at Froude 0.28 [-] is computed and analysed in fixed-dynamic position. The vessel is initially fixed at the dynamic position ($-1.82L_{pp} \times 10^{-3} [-]$ in sinkage and 0.108 [deg] in trim by bow), and without allowing to have sinkage and trim during the computation. The calculation is performed on 12 cores. The flow time is set as 20 [sec]. The model parameters are as follows:

Model speed, V_m	2.097	[m/sec]
Froude Number, F_n	0.28	[-]
Reynolds' Number, Re	1.195E+07	[-]
Test Condition	<ul style="list-style-type: none"> Bare Hull Calm Water Fixed Dynamic Condition (sinkage and trim imposed) 	

Table 4-3 DTMB Fixed Condition Settings

The other key information have been already introduced in Section 3.1/

Presentation and discussion of the results

The computation was performed with 12 cores for the flow time of 20 [sec]. It took 26.3 [hr] (315.60 [core-hr]). The viscous, pressure and total forces of half-body in the longitudinal direction are plotted in Figure 4-6. In viscous force, convergence is achieved after 1000 iterations (10 [sec]). In pressure force, the convergence can also be seen after 1600 iterations with slight oscillations.

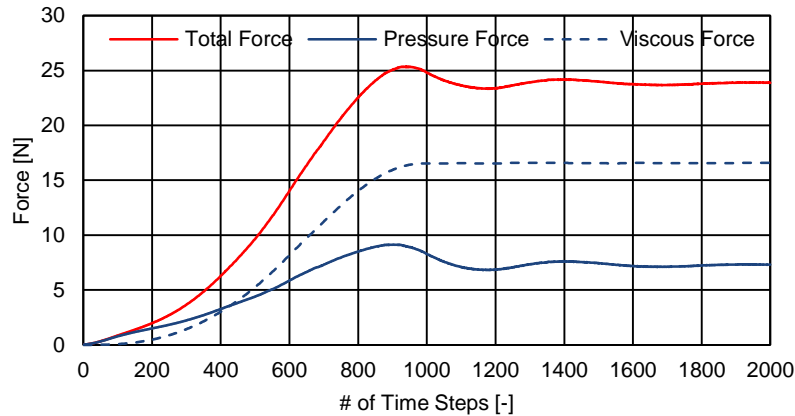


Figure 4-6 Convergence History of Forces in DTMB Fixed Condition

In Table 4-4, the resistance coefficients are computed and analysed. In general, the resistance coefficients of *foamStar* are found to be slightly over-predicted.

	EFD (INSEAN)	<i>foamStar</i>	Difference [%]
$C_f [-] \times 10^3$	2.91	3.105	6.70
$C_r [-] \times 10^3$	1.32	1.355	2.63
$C_t [-] \times 10^3$	4.23	4.460	5.43

Table 4-4 Comparison of Resistance Coefficients in DTMB Fixed Condition

It is observed that there are 2.63 [%] difference in residuary resistance coefficient, 6.70 [%] in frictional resistance coefficient and overall 5.43 [%] in total resistance coefficient in comparison with INSEAN results.

The obtained free surface pattern and wave contours are compared with INSEAN results in Figure 4-7. In the given experimental data, the transverse cut starts from $y/L_{pp} = 0.08 [-]$ and the bow wave field is not well present. Hence, the visual scale of *foamStar* results is adjusted to be in range of the experimental data. It is obvious that the wave pattern and elevation contour values in the far field locations well agree to each other.

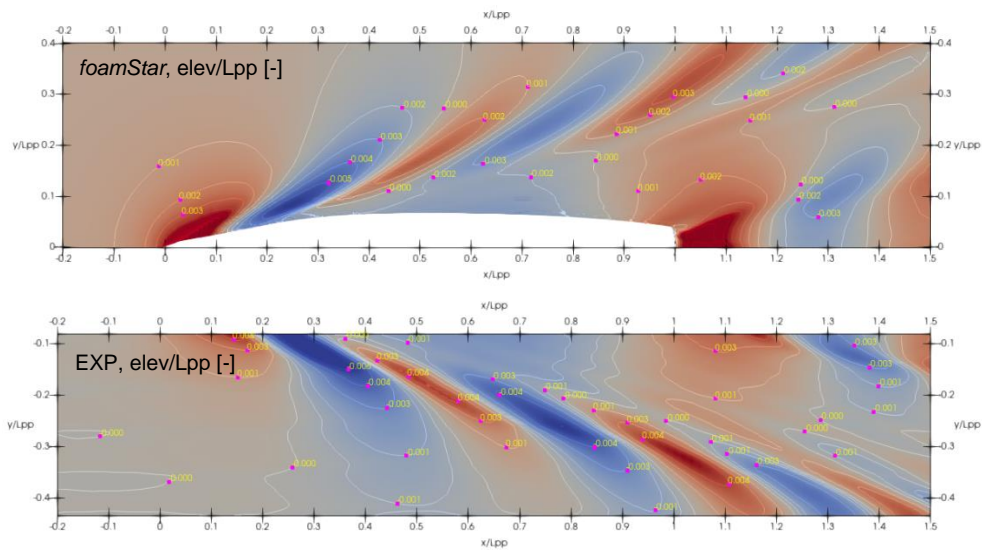


Figure 4-7 Normalised Free Surface Elevation: Comparison with Experimental Results

The longitudinal wave profiles at three transverse locations: $y/L_{pp} = 0.082, 0.172$ and 0.301 [-] are plotted after global view of the wave pattern and elevation contour are illustrated. The wave profiles along these cuts are analysed and compared with INSEAN results.

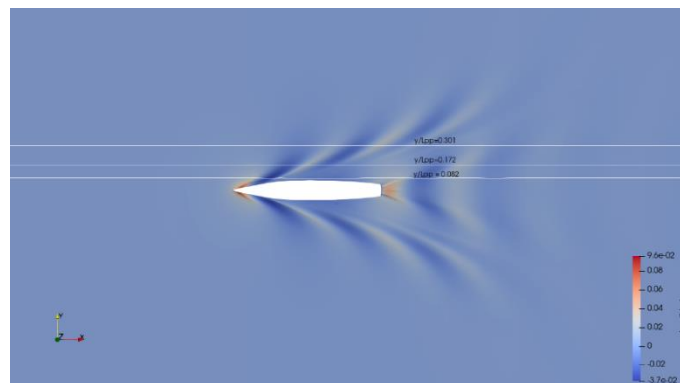
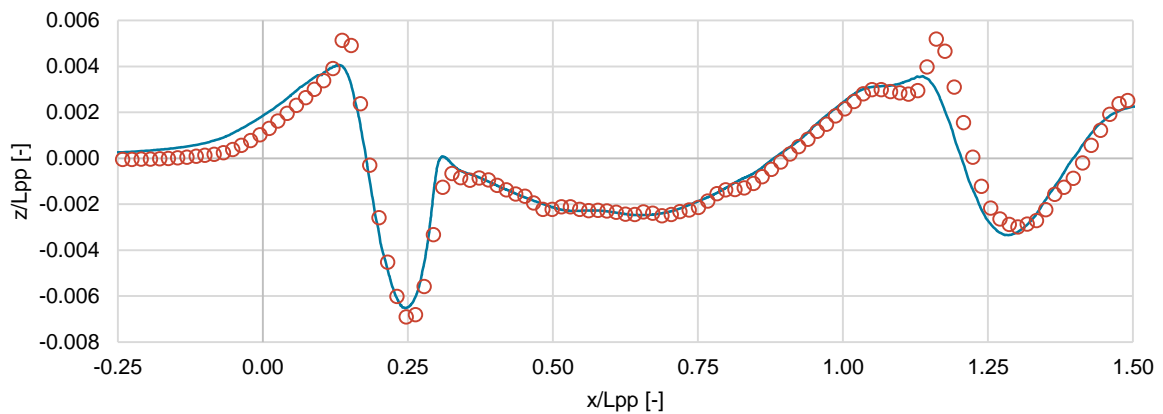
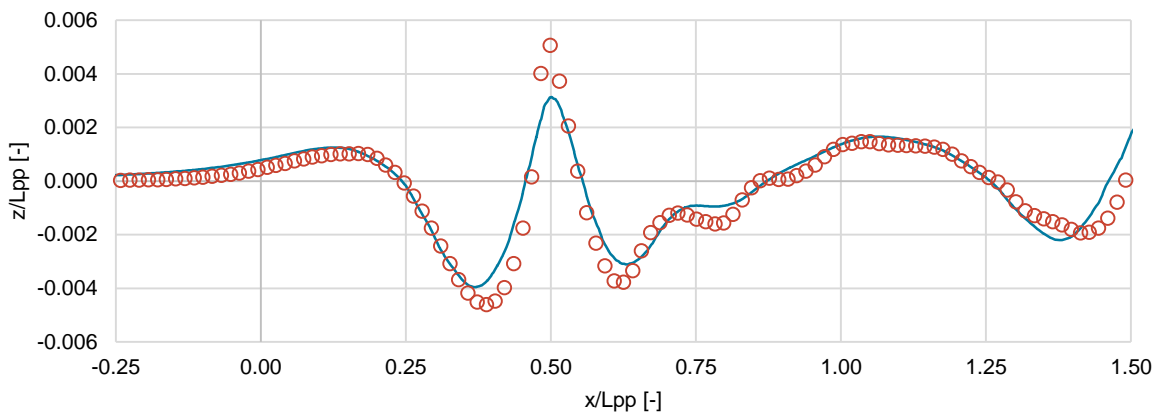


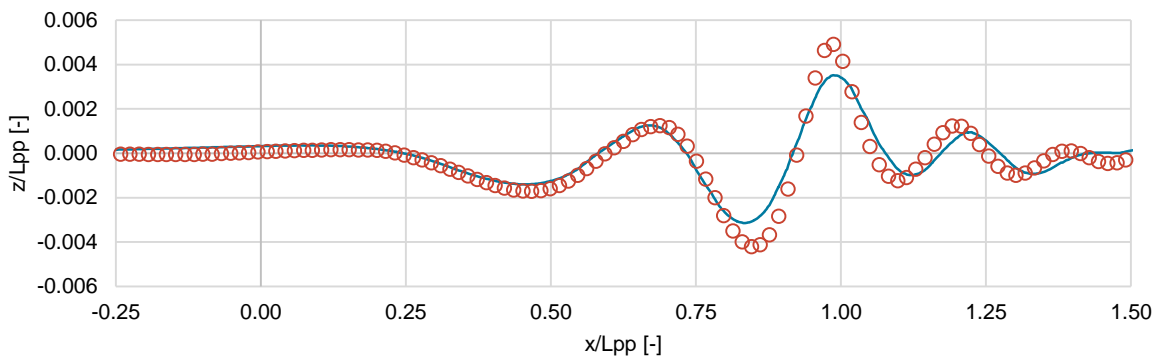
Figure 4-8 Longitudinal Cut Locations in Far Field from Hull



(a)



(b)

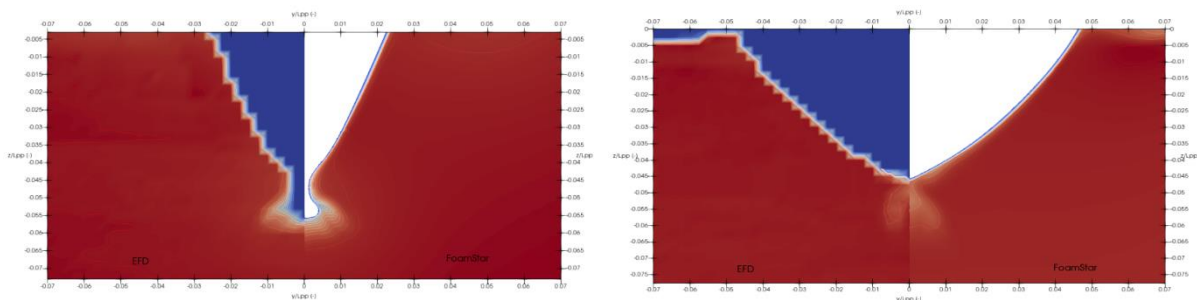


(c)

Figure 4-9 Wave Profiles at (a) $y/L_{pp} = 0.082$ [-], (b) $y/L_{pp} = 0.172$ [-] and (c) $y/L_{pp} = 0.301$ [-]

With reference to Figure 4-9, it can be observed that *foamStar* can capture well the wave elevation in most locations along the cut generally. But at crest and trough locations, the predicted wave is neither high nor low enough. At $y/L_{pp} = 0.301$ [-] location, the wave profile is identical until $x/L_{pp} = 0.65$ [-], then the phase-shifting occurs after the stern.

Overall, the computational longitudinal velocity field and the contour results quite agree with the experimental results. The comparisons at different sections are shown in Figure 4-10.



(a) $x/L_{pp} = 0.1$ [-]

(b) $x/L_{pp} = 0.2$ [-]

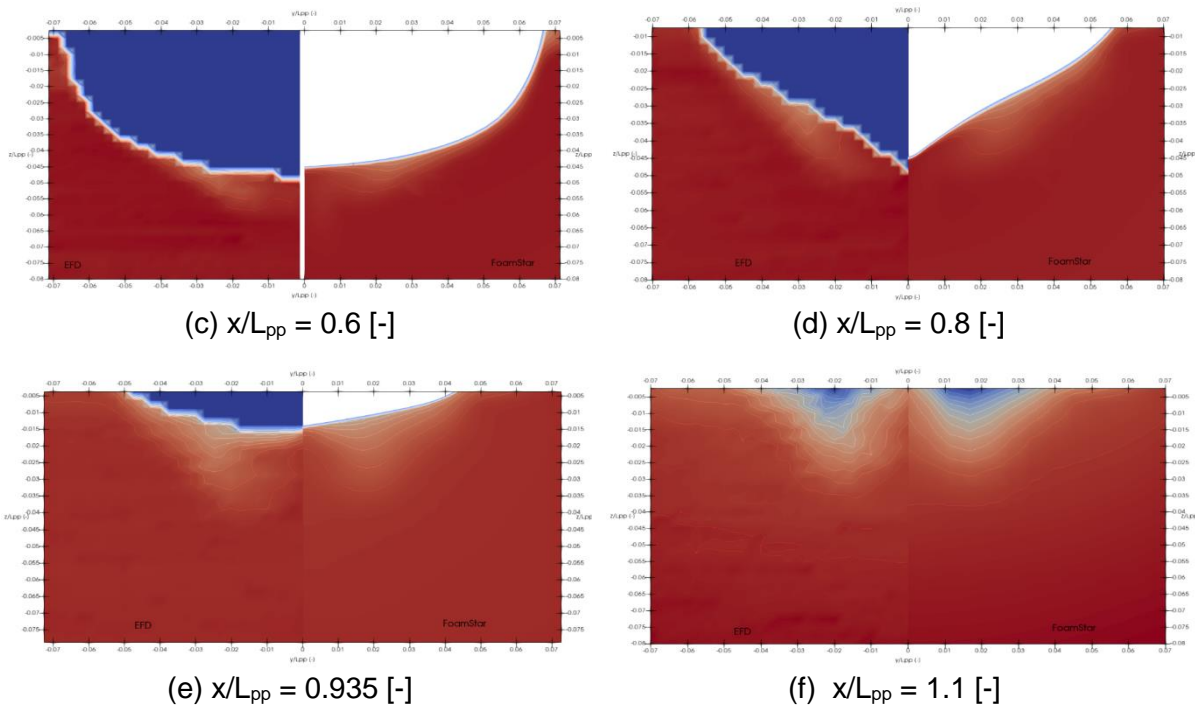


Figure 4-10 Velocity Field and Velocity Contour at Different Locations along Hull

Since the vessel has the peculiar shape of bulb and due to the presence of the sonar dome, bow bilge vortices are generated (Olivieri, Pistani, Avanzini, F. Stern, & Penna, 2001). By comparing *foamStar* results with the experimental ones, this bow bilge vortex can be seen clearly at $x/L_{pp} = 0.2$ [-]. The second vortex stern bilge vortex is expected at $x/L_{pp} = 0.6$ [-]. At $x/L_{pp} = 0.8$ [-], no vortices are observed but at $x/L_{pp} = 1.1$ [-], there is one vortex occurred. The flow phenomena are well in line with the reference.

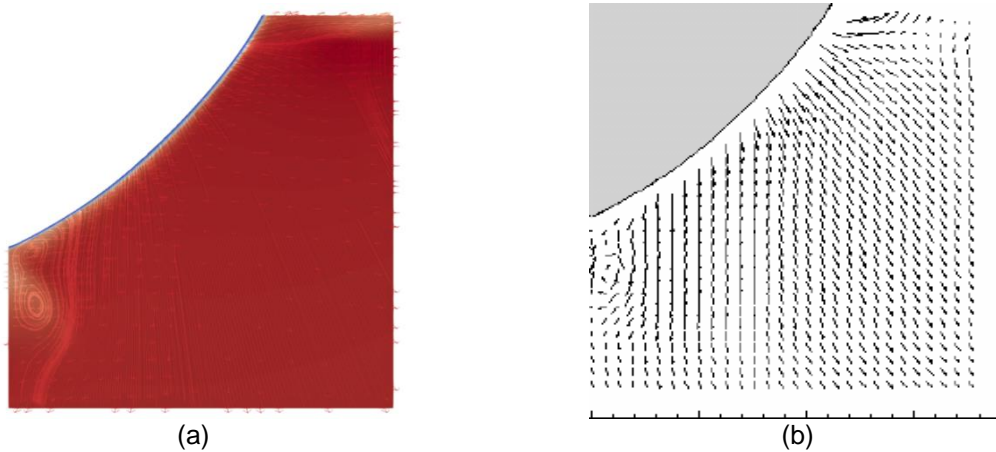


Figure 4-11 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.2$ [-] (a) *foamStar* and (b) Experiment

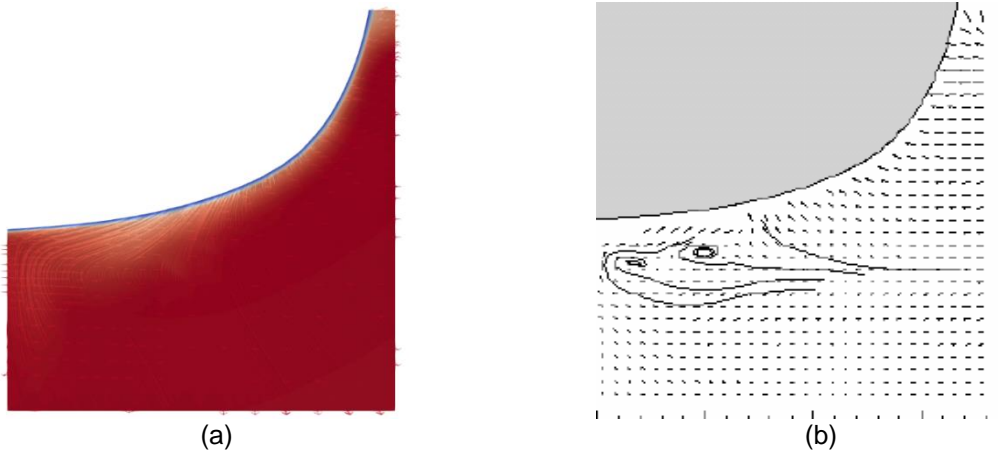


Figure 4-12 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.6$ [-] (a) foamStar and (b) Experiment

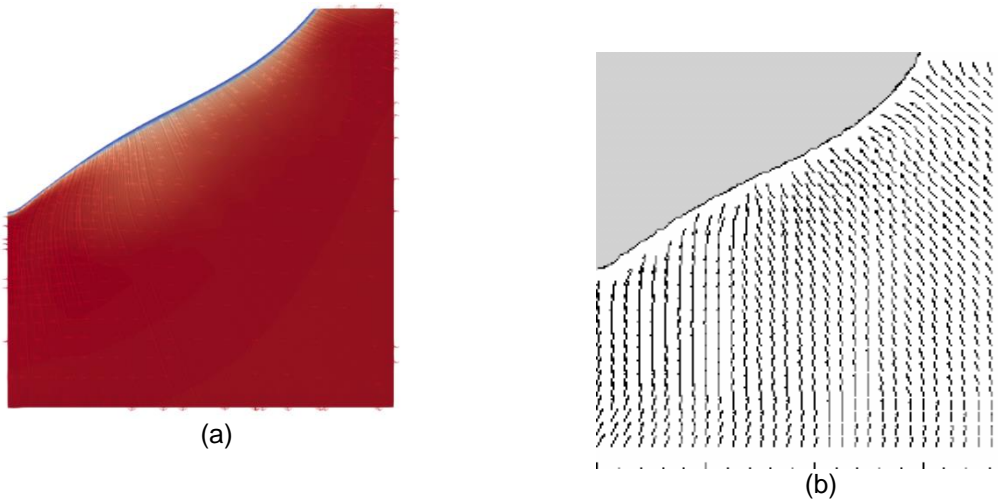


Figure 4-13 Streamline and Cross-flow Vectors at $x/L_{pp} = 0.8$ [-] (a) foamStar and (b) Experiment

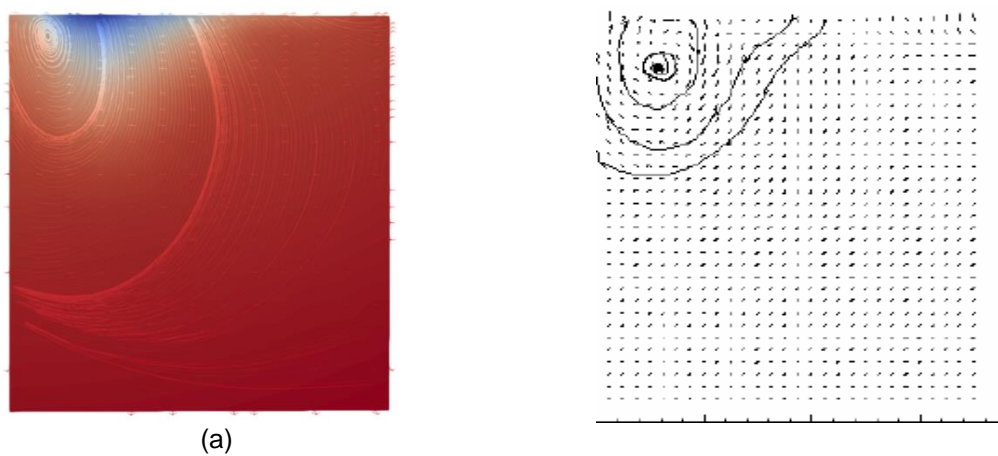


Figure 4-14 Streamline and Cross-flow Vectors at $x/L_{pp} = 1.1$ [-] (a) foamStar and (b) Experiment

4.3/ DTMB 5415 Combatant in Free Condition

The resistance, sinkage and trim of DTMB 5415 bare hull in calm water condition for Froude range of 0.10 – 0.45 [-] are computed and compared with experimental results. Contrary to previous case, the model is initially at even keel position, and allowed to have sinkage and trim freely during the computations. The calculations are performed with 30 cores.

Froude Number, F_n	0.10 – 0.45	[-]
Reynolds' Number, Re	4.27E+06 - 1.92E+07	[-]
Test Condition	<ul style="list-style-type: none"> • Bare Hull • Calm Water • Still Water Position (Free sinkage and trim) 	

Table 4-5 DTMB in Free Condition Settings

4.3.1/ Resistance and Resistance Coefficients

The total drag forces are presented hereafter and compared to experimental results. In Figure 4-15, it can be seen that the trend of two resistance curves are quite similar to each other. The total resistance increases as the speed is increased. The discrepancy in total resistance increases up to 8.10 [%] until $F_n=0.30$ [-], then it decreases rapidly to 2.49 [%] as in Table 4-6.

Fn [-]	Total Resistance		
	EFD [N]	foamStar [N]	Difference [N]
0.10	5.47	5.80	0.33 (6.03 [%])
0.14	10.34	11.07	0.73 (7.06 [%])
0.20	21.58	23.21	1.63 (7.55 [%])
0.28	45.18	48.51	3.33 (7.37 [%])
0.30	54.31	58.71	4.40 (8.10 [%])
0.40	138.35	143.05	4.70 (3.40 [%])
0.41	152.70	158.81	6.11 (4.00 [%])
0.45	216.33	221.71	5.38 (2.49 [%])

Table 4-6 Total Force of foamStar and Experiment in DTMB Free Condition

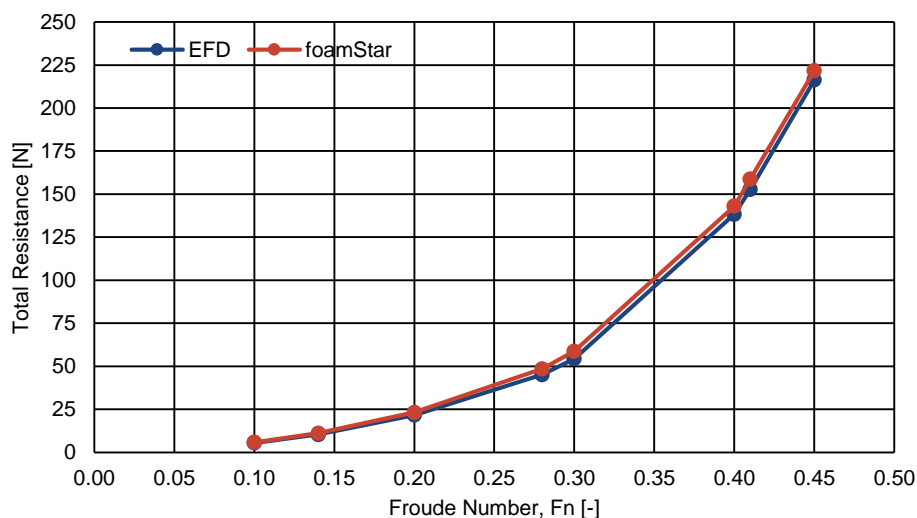


Figure 4-15 Comparison of Total Force in DTMB Free Condition

The total, viscous and pressure components of *foamStar* are then converted into coefficients.

Fn [-]	EFD			<i>foamStar</i>		
	$C_t [-] \times 10^3$	$C_f [-] \times 10^3$	$C_r [-] \times 10^3$	$C_t [-] \times 10^3$	$C_f [-] \times 10^3$	$C_r [-] \times 10^3$
0.10	4.020	3.500	0.523	4.266 (+6.12 [%])	3.662 (+4.62 [%])	0.604 (+15.51 [%])
0.14	3.890	3.290	0.605	4.173 (+7.28 [%])	3.448 (+4.82 [%])	0.725 (+19.77 [%])
0.20	3.950	3.080	0.864	4.248 (+7.55 [%])	3.272 (+6.23 [%])	0.976 (+12.98 [%])
0.28	4.230	2.910	1.320	4.548 (+7.52 [%])	3.124 (+7.35 [%])	1.424 (+7.89 [%])
0.30	4.430	2.880	1.560	4.794 (+8.22 [%])	3.095 (+7.47 [%])	1.699 (+8.91 [%])
0.40	6.350	2.740	3.610	6.571 (+3.48 [%])	3.040 (+10.93 [%])	3.531 (-2.18 [%])
0.41	6.670	2.730	3.940	6.943 (+4.09 [%])	3.034 (+11.13 [%])	3.909 (-0.79 [%])
0.45	7.850	2.690	5.120	8.049 (+2.54 [%])	3.003 (+11.62 [%])	5.046 (-1.44 [%])

Table 4-7 Resistance Coefficients and Percent Differences in DTMB Free Condition

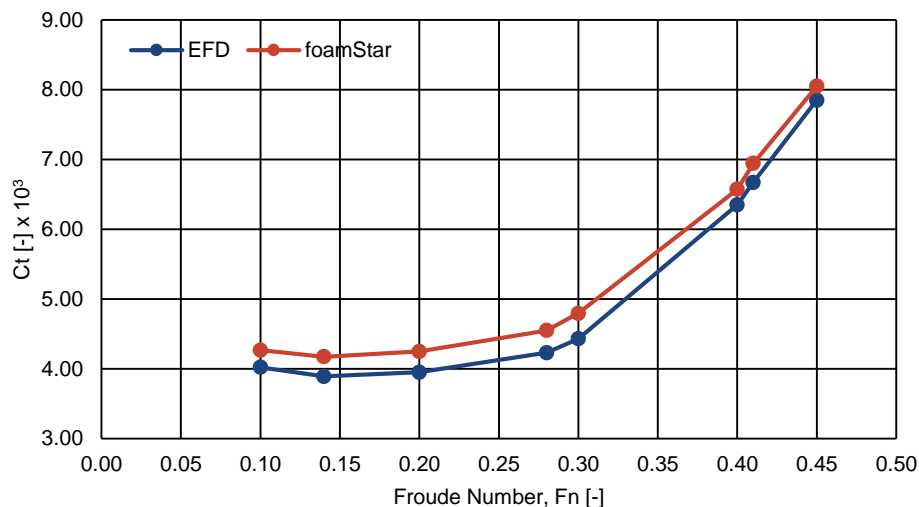


Figure 4-16 Comparison of Total Resistance Coefficient in DTMB Free Condition

According to Table 4-7, the difference in residuary resistance decreases when the speed increases. It can be said that the pressure resistance can be well captured starting from $Fn = 0.40 [-]$. The difference in total resistance coefficient also decreases after $Fn = 0.30 [-]$. Between $0.10 [-]$ and $0.30 [-]$, the discrepancies increase. The average discrepancies of 5.85 [%] is found in total resistance coefficient.

4.3.2/ Sinkage and Trim

The heave and pitch motions from *foamStar* are extracted and compared with experimental results as sinkage and trim. The heave motion is non-dimensionalised by the model length.

Fn [-]	Sinkage/ $L_{pp} \times 10^{-3}$ [-]			Trim [deg]		
	EFD	<i>foamStar</i>	Difference [%]	EFD	<i>foamStar</i>	Difference [%]
0.10	0.1740	0.1858	6.77	0.018	0.013	-28.78
0.14	0.5040	0.3635	-27.88	0.028	0.025	-10.18
0.20	0.9320	0.7958	-14.61	0.061	0.056	-8.09
0.28	1.8200	1.7555	-3.54	0.108	0.097	-10.62
0.30	2.2300	2.0938	-6.11	0.093	0.078	-15.62
0.40	4.4600	4.2057	-5.70	-0.333	-0.300	-9.90
0.41	4.7000	4.4407	-5.52	-0.421	-0.434	3.01
0.45	5.6200	4.9720	-11.53	-1.009	-0.998	-1.09

Table 4-8 Sinkage and Trim Results and Comparison in DTMB Free Condition

The sinkage results are plotted as in Figure 4-17. *foamStar* and experimental results remain quite consistent until $Fn = 0.41$ [-] but the gap increases beyond $Fn = 0.41$ [-].

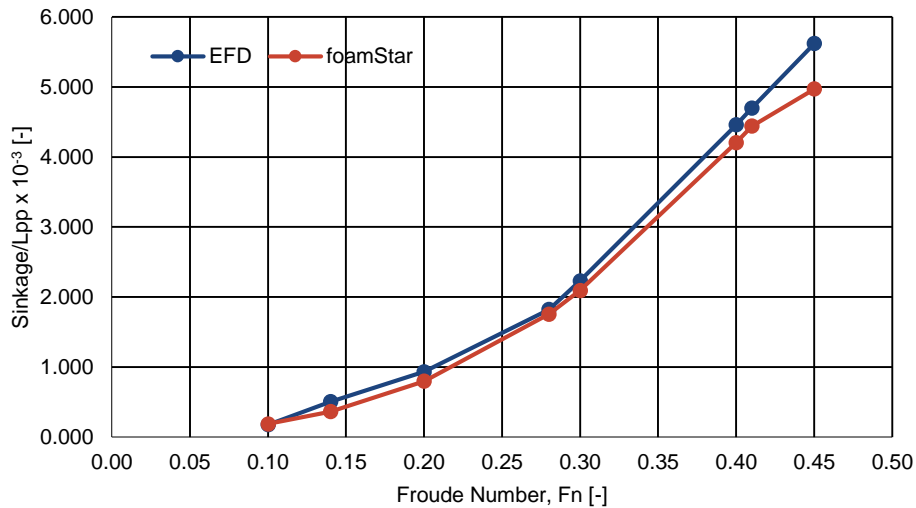


Figure 4-17 Comparison of Non-dimensionalised Sinkage Results in DTMB Free Condition

For the trim results, the model experienced bow trim until $Fn = 0.30$ [-] and from this point, the model becomes trimming by stern. The nature and tendencies of the trim result curves are quite matching between EFD and CFD results as illustrated in Figure 4-18.

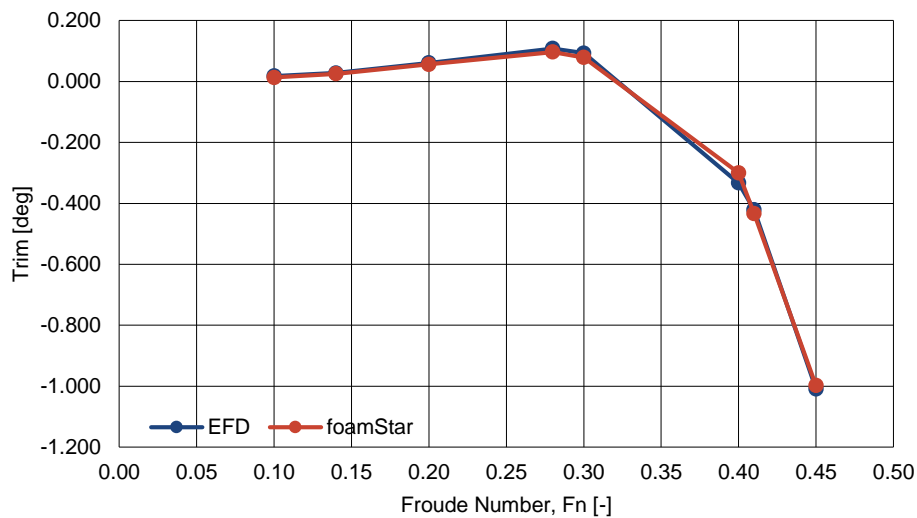


Figure 4-18 Comparison of Trim Results in DTMB Free Condition

Overall, it can be concluded that the results from *foamStar* in KCS and DTMB cases are quite good in comparison with the experimental results. The resistance coefficients are well predicted with a small tendency to over-estimation. The flow phenomena can be captured quite correctly. These are encouraging results for future use of the CFD tool. However, additional validation cases should be run for full validation.

5 / Industrialisation of *foamStar* – Script Development & Validation

This section presents the work carried out on script development in view of automating the setup and launching series of computation for industrial applications with *foamStar*. These types of scripts are practically useful for studies of ship optimisation where multiple hull forms are analysed repeatedly. The script already exists in BVS for ISIS-CFD, the objective is to adapt it to *foamStar* specifications while keeping a structure as close as possible to the existing one.

First, the architecture of the adapted script will be presented. Each part included in the routine of industrial automation will be explained. Finally, the script is tested on an industrial hull form optimisation study of BV Solutions for validation purposes.

6 / Conclusion

The work performed in this internship enabled to verify the robustness and sensitivity of *foamStar* in view of its industrial use on resistance cases. At first, multiple sensitivity studies both on numerical parameters and grid parameters were conducted. It showed the importance of good practices to optimise its use and highlighted some of the key setup options to be used. Moreover, some unstable cases/configurations were encountered showing the poor stability of *FoamStar*, especially with regard to the mesh.

Then, the benchmark cases of KCS and DTMB 5415 hulls have been performed in calm water resistance computations. Good agreement was observed in resistance coefficients, free surface pattern and wave profiles between *foamStar* and experimental results. It can be said that *foamStar* can capture flow phenomena with the certain degree of accuracy. These results are encouraging and tend to validate the capability of the software for resistance predictions with a small tendency to over-estimate it.

For industrialisation purpose, the setup procedure to run *foamStar* calculation have also been automatized by modifying the BVS's existing script. The workability and the reliability is well established by testing it on hull optimisation cases. In addition, the results obtained for this optimisation study showed that while over-estimating the resistance, *foamStar* results are in line with ISIS-CFD results with regard to relative comparison of the various hull forms tested. It is a positive outcome which tends to validate its use for such applications. However, it needs to be investigated further prior to any industrial use as one case is not enough for validation.

As a conclusion, the work performed in this internship showed promising results in view of *foamStar* use for resistance applications but additional work remains necessary to secure its use on industrial cases.

7 / Future Work

The typical grid with viscous layer generated from Hexpress has good quality and working well for the particular solver such as ISIS-CFD but that same grid has high aspect ratio, skewness and non-orthogonality cells when it is used in *foamStar*/OpenFOAM. The current viscous layer insertion technique has the significant impact on the grid quality, which increases CPU costs drastically.

Throughout the internship, the studied cases are mainly of calm water condition. The capability of *foamStar* in wave condition has not been verified yet. The computations to estimate the added resistance of the hull in wave condition shall be the next step. One other interesting idea would be to study the performance of *foamStar* using steady solver.

Since *foamStar* is developed based on open-source CFD packages, the numerical parameters can be modified by experienced users. According to robustness and sensitivity analysis results, *foamStar* can still be optimised especially for the computation cost and stability to mesh. Development work to improve these points would be very interesting.

The script developed for industrial application has been verified with hull optimisation cases in calm water condition. The same script can also be applied for the wave condition with the additional inputs in the parameter settings but it needs to be verified.

8 / References

- Asnaghi, A., Feymark, A., & Bensow, R. E. (2015). Computational Analysis of Cavitating Marine Propeller Performance using OpenFOAM. *Fourth International Symposium on Marine Propulsors smp'15*. Austin.
- Bruinsma, N. (2016). *Validation and Application of a Fully Nonlinear Numerical Wave Tank*. Delft: TU Delft.
- Gaggero, S., Villa, D., Tani, G., Viviani, M., & Bertetta, D. (2017). Design of ducted propeller nozzles through a RANSE-based optimization approach. *Ocean Engineering*, 444-463.
- Gentaz, L. (2020). *NUMHY-Viscous Flow CFD*. Nantes: Ecole Centrale de Nantes.
- GREENFIELDS, C. (2017, March 2). *OpenFOAM v5 User Guide: 5.4 Mesh generation with snappyHexMesh*. Retrieved March 31, 2021, from CFD Direct: <https://cfdirect.com/openfoam/user-guide/v5-snappyHexMesh/#x26-1910005.4>
- GREENSHIELDS, C. (2017, March 2). *OpenFOAM v5 User Guide: 1 Introduction*. Retrieved March 31, 2021, from CFD Direct: <https://cfdirect.com/openfoam/user-guide/v5-introduction/#x3-20001>
- GREENSHIELDS, C. (2017, March 2). *OpenFOAM v5 User Guide: 3.5 Standard solvers*. Retrieved March 31, 2021, from CFD Direct: <https://cfdirect.com/openfoam/user-guide/v5-standard-solvers/#x13-1000003.5.4>
- GREENSHIELDS, C. (2017, March 2). *OpenFOAM v5 User Guide: 5.3 Mesh generation with blockMesh*. Retrieved March 31, 2021, from CFD Direct: <https://cfdirect.com/openfoam/user-guide/v5-blockMesh/#x25-1790005.3>
- ITTC. (1996). *CFD, Resistance and Flow Benchmark Database for CFD Validation for Resistance and Propulsion*. International Towing Tank Committee (ITTC).
- Jasak, Hrvoje; Tukovic, Zeljko. (2010). Dynamic Mesh Handling in OpenFOAM applied to Fluid-Structure Interaction Simulations. *European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010*, (pp. 1-16). Lisbon.
- Kim, W. J., Van, S. H., & Kim, D. H. (2001). Measurement of Flows around Modern Commercial Ship Models. *Experiments in Fluids*, 567-578.
- Le Touze, D. (2020). *Introduction to the Simulation of Turbulence*. Nantes, France.
- LI, Z. (2018). *Two-Phase Spectral Wave Explicit Navier-Stokes Equations Method for the Wave-Structure Interactions*. Nantes: Ecole Centrale de Nantes.
- Li, Z., Deng, G., Queutey, P., Bouscasse, B., Ducrozet, G., Gentaz, L., . . . Ferrant, P. (2019). Comparison of wave modeling methods in CFD solvers for ocean engineering applications. *Ocean Engineering*, 188. doi:<https://doi.org/10.1016/j.oceaneng.2019.106237>
- Liu, C. (2019). *Prediction of Forces and Moments in Multiphase Naval Simulations*. Waterloo: University of Waterloo.
- Monroy, Charles; Seng, Sopheak; Malenica, Sime. (2016). Developpements et Validation de l'Outil CFD OpenFOAM pour le Calcul de Tenue a la Mer. *15emes Journees de L'Hydrodynamique*. Brest.
- Olivieri, A., Pistani, F., Avanzini, A., F.Stern, & Penna, R. (2001). *Towing Tank Experiments of Resistance, Sinkage and Trim, Boundary Layer, Wake, and Free Surface Flow around a Naval Combatant INSEAN 2340 Model*. IOWA: University of Iowa.
- Shahzad, M. T. (2017). *Ship Resistance Computations using OpenFOAM*. Szczecin: West Pomeranian University of Technology, Szczecin.
- Tober, H. (2020). *Evaluation of Drag Estimation Methods for Ship Hulls*. Stockholm: KTH Royal Institute of Technology.
- Turunen, T., Siikonen, T., Lundberg, J., & Bensow, R. (2014). Open-Water Computations of a Marine Propeller using OpenFOAM. *6th European Conference on Computational Fluid Dynamics (ECFD VI)*, (pp. 1123-1134). Barcelona.
- Vukcevic, V., & Jasak, H. (2014). OpenFOAM in Marine Hydrodynamics. *21st symposium Sorta 2014*, (pp. 369-376). Sorta.
- Zhaobin Li, G. D., Queutey, P., Bouscasse, B., Ducrozet, G., Gentaz, L., Touzé, D. L., & Ferrant, P. (2019). Comparison of wave modeling methods in CFD solvers for ocean engineering applications. *Ocean Engineering*, 188. doi:<https://doi.org/10.1016/j.oceaneng.2019.106237>