



SHAPE pilot Monotricat SRL: Hull resistance simulations for an innovative hull using OpenFOAM

Lilit Axner^{a,b}, Jing Gong^{a,b,*}, Alessandro Chiarini^c, Luigi Mascellaro^d

^a*PDC-HPC, KTH Royal Institute of Technology, Stockholm, SE-10044, Sweden*

^b*The Swedish e-Science Center (SeRC), KTH Royal Institute of Technology, Stockholm, SE-10044, Sweden*

^c*Super Computing Solutions, CINECA, Bologna, 40033, Italy*

^d*Monotricat, Via Fosso della Castelluccia 146/22, Roma, 00134, Italy*

Abstract

In the project we conduct a whole hull simulation of a ship, (including mesh generation of complex geometries, efficient solvers with various turbulent modelling and optimized parameters as well as visualization), along with a performance analysis for the simulation. This project makes it possible for Monotricat to take advantage of HPC-enabled simulation tools and potentially replace the traditional methods.

1. Introduction

Monotricat SRL [1] is an Italian company that designs an innovative type of hull (MONOTRICAT), which is characterized by being highly efficient in hydrodynamic terms compared to other types of hulls. The aim of Monotricat's hull architecture is to obtain higher efficiency and larger load capacity. However the design and development processes for new hulls are very expensive and time consuming. In order to obtain extensive measurements relating to the performance of a new or improved hull, a robust and thorough test phase is required. For the MONOTRICAT hull, tests were run in the naval basins of Rome (CNR-INSEAN) and Trieste using a large number of models, and thus with a large expenditure of time and effort, and with resulting high costs.

There are many advantages to using Computational Fluid Dynamics (CFD) in the first phase when designing a hull or other item, and consequently CFD has been widely accepted as a simulation and optimization tool in many industrial branches. When it comes to designing hulls, the most important aspects of using CFD techniques are the ability to handle complex geometry with relevant details, the efficient simulation process (from geometry to solution, parametric studies, optimization studies, user interface); the adequate modelling of turbulence, free-surface effects and cavitation, and the coupled simulation of flow and flow-induced motion.

The aim of this particular PRACE SHAPE project is to develop accurate and reliable CFD tools that Monotricat can use to replace the highly expensive field tests. Moreover the project has produced a tool that can be used to better understand the hydrodynamic behaviour of hulls and that will also provide a quick, easily accessible and scalable HPC-based service that will optimize the research and development costs for the company. In this project we conduct not only the performance analysis but also whole hull simulation processing, e.g. mesh generation of complex geometries, efficient solvers with various turbulent modelling and optimized parameters

* Corresponding author. *E-mail address*: gongjing@kth.se

as well as visualization. This project creates a possibility to assist Monotricat to take advantage of HPC enabled simulation tools and potentially replace the traditional methods.

The open source code OpenFOAM [2] was chosen for this project because it is an open source package that is widely accepted by the CFD community, both in academia and industry. In this software package, *snappyHexMesh* [3] is employed to generate three-dimensional meshes of hexahedra from triangulated surface geometries in Stereolithography (STL) format. The *interFoam* solver is used to solve the two incompressible fluids (water and air) using a VOF (volume of fluid) phase-faction-based interface capturing approach. OpenFOAM also provides the post-processing tool *paraFoam* for visualizing the results.

2. Theoretical background

2.1 The Volume of fluid method

The Reynolds averaged momentum equation is written as

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot UU = -\nabla P + \rho g + \nabla \cdot \left[(\mu + \mu_t)(\nabla U + \nabla U)^T \right]$$

where U is the velocity, P is the pressure, g is the gravity, ρ is the density and μ is the dynamic viscosity [4]. The eddy viscosity μ_t is a scalar property calculated from the two transported variables in the two-equation turbulent models.

There are two main methods to compute the free surface, the interface tracking method and the front capturing methods [5]. In this project, the VOF method is used to implement the front capturing method for the free surface. The VOF method is a two-phase surface compression method that solves the Navier-Stokes equations and an additional advection transport equation for the volume fraction or scalar indicator function [9], [6], and [14]. The scalar function $\alpha(x,t) \in [0,1]$ represents the phase of the fluid in each cell. For the hull simulation, $\alpha = 1$ indicates that water fills the whole cell, while $\alpha = 0$ means that the cell is filled with air.

$$\mu(x,t) = \mu_{water}\alpha + \mu_{air}(1-\alpha)$$

$$\rho(x,t) = \rho_{water}\alpha + \rho_{air}(1-\alpha)$$

The α function can be calculated by a separated transport equation:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot U\alpha = 0$$

OpenFOAM introduces an extra artificial compression term into the equation and the modified governing equation for the α function can be written as

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot U\alpha + \nabla \cdot W[\alpha(1-\alpha)] = 0$$

where W is an artificial velocity field [4].

2.2 The turbulent models

The two-equation turbulent models, the $k-\varepsilon$ model and the $k-\omega$ model, are the most commonly used in naval architecture. Note that k is the turbulent kinetic energy per unit mass ε is the turbulent dissipation and ω is the specific rate of dissipation. However, neither the $k-\varepsilon$ model nor the $k-\omega$ model is suitable for flows with strong stream-wise vortices [9]. In his articles [10,11] Menter proposed that both models should be used together, at the hull surface and in the free stream outside the boundary layer. In [10] and [11] Menter suggested using both the $k-\varepsilon$ and the $k-\omega$ models together to improve prediction of the principal shear stresses, which is known as the $k-\omega$ SST model. When calculating the coefficients for the turbulence models, the approximate coefficients of the $k-\omega$ SST model can be guided, as described in [12] and [13].

2.3 Boundary conditions

The boundary conditions used in the project are illustrated in Figure 1.

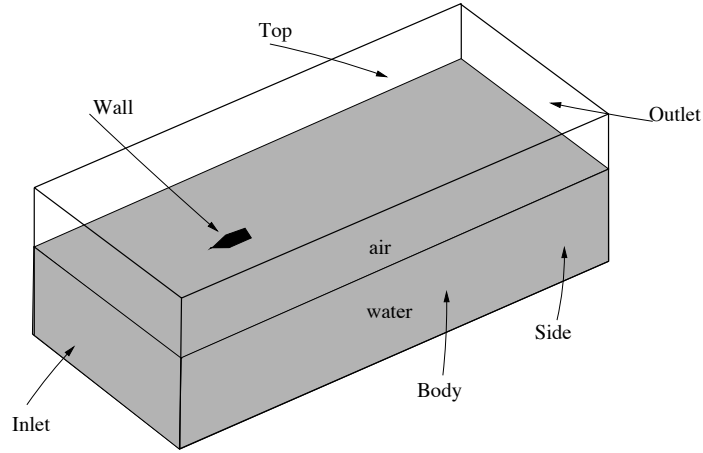


Figure 1: The boundary conditions

The boundary conditions are

$$\text{Top: } \frac{\partial U}{\partial n} = 0, \quad p = 0, \quad \alpha = 0$$

$$\text{Hull: } U = 0, \quad \frac{\partial p}{\partial n} = 0, \quad \frac{\partial \alpha}{\partial n} = 0$$

$$\text{Inlet: } U = U_{\infty}, \quad \frac{\partial p}{\partial n} = 0, \quad \alpha = \begin{cases} 1 & \text{if } z < 0 \\ 0 & \text{otherwise} \end{cases}$$

$$\text{Outlet: } \frac{\partial U}{\partial n} = 0, \quad p = 0, \quad \frac{\partial \alpha}{\partial n} = 0$$

Where n is the outer normal vector.

3. OpenFOAM

3.1. *interFoam* solver

In this project the multiphase flow solver is of interest since it uses the VOF method to solve the free surface for two incompressible flows. *interFoam* is an OpenFOAM solver for two incompressible fluids capturing the interface using the VOF methods and PISO (“Pressure Implicit with Splitting of Operators”) for pressure velocity coupling [1]. When using explicit difference schemes to solve partial differential equations, a correct value of the Courant number (CFL) is essential to obtain convergence and stability of the solution [14].

3.2. Mesh generation

The embedded OpenFOAM mesh generation application *snappyHexMesh* was used to build the mesh for the simulation. The basic purpose of this application is to build a custom mesh with several refinement regions [1]. We assume that the first mesh (referred to as the 0-level mesh) has been obtained with a mesh generator and that this mesh has been generated with an a priori knowledge of the problem, for example with boundary layer elements near the wall. The mesh for the hull simulation is shown in Figure 2.

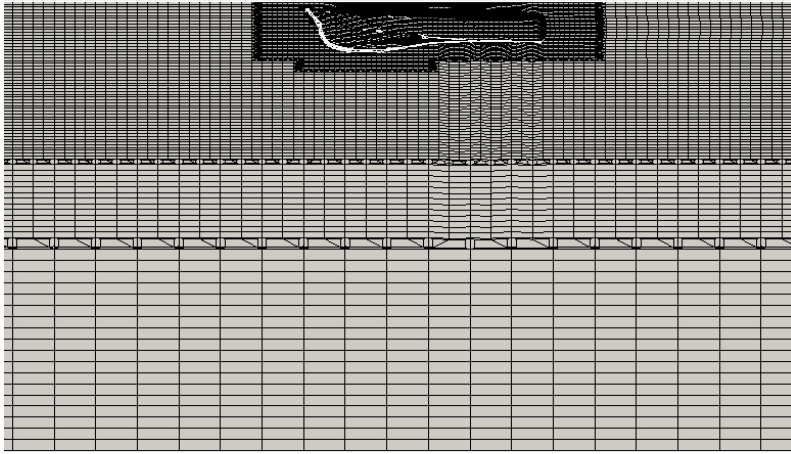


Figure 2: The mesh for hull simulations

4. Results

All the tests were conducted on the PRACE Tier-1 system Lindgren at KTH-PDC. Lindgren is a Cray XE6 system; based on AMD Opteron 12-core Magny-Cours processors and Cray Gemini interconnect technology. It consists of 1516 compute nodes with a theoretical peak performance of 305 TFlop/s. OpenFOAM v2.2.0 built with the GNU compiler v4.6.3 was used in the project.

The velocity field on the air/water interface surface with $\alpha = 0.5$ is shown in Figure 3.

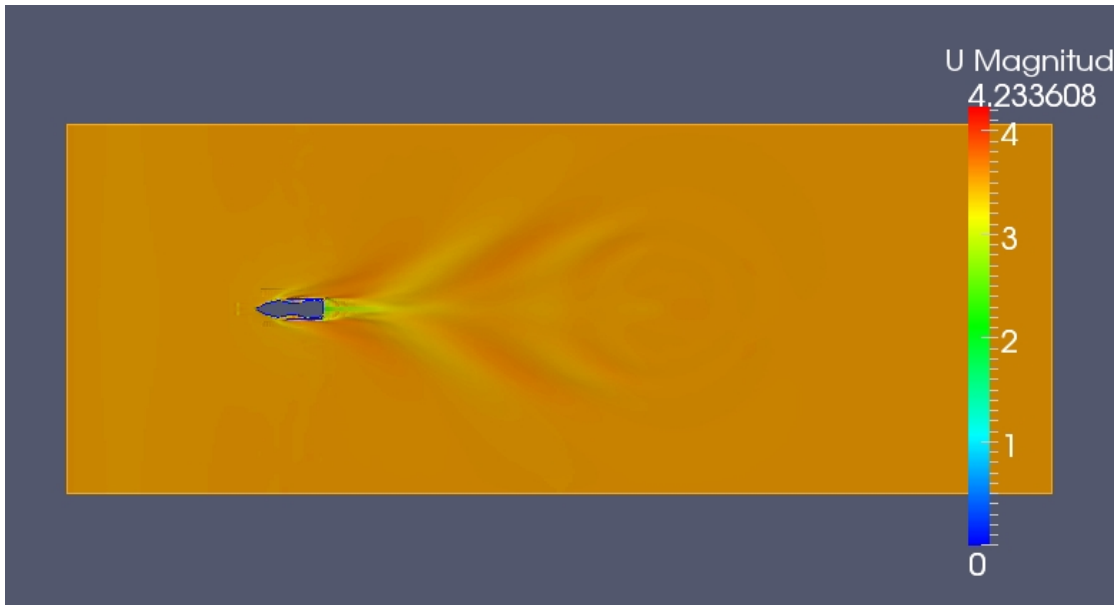


Figure 3: The velocity field on the air/water interface surface

4.1. Hull resistance

The volumetric Froude number is defined as

$$F_{nv} = \frac{U}{\sqrt{g(Vol)^{1/3}}}$$

where U is the velocity, g is the gravitational constant, and Vol is the volume of the ship. With different values of the Froude number, the hydrodynamic behaviour is different. The experimental ship resistances at various volumetric Froude numbers are shown in Table 1.

Volumetric Froude Number (F_{nv})	Velocity (m/s)	Resistance (N)
0.813	1.766	39.338
0.975	2.117	64.746
1.137	2.470	84.562
1.303	2.832	97.315
1.462	3.176	105.752
1.625	3.529	115.366
1.788	3.882	129.198
2.113	4.588	164.612
2.275	4.941	192.472
2.438	5.294	228.671

Table 1: The experimental resistances

Figures 4(a)-(j) compares the numerical results obtained using OpenFOAM with the experimentally derived curve for different values of F_{nv} between 0.813 and 2.438. For all cases, convergence is reached after 4-6 seconds.

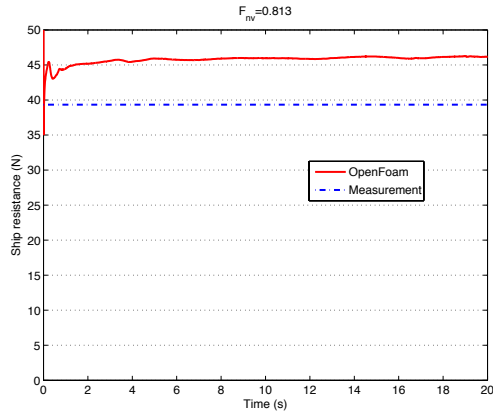


Figure 4(a): The ship resistance at $F_{nv} = 0.813$

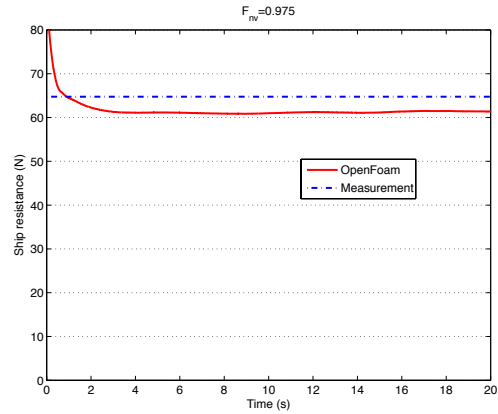


Figure 4(b): The ship resistance at $F_{nv} = 0.975$

From these figures, it is clear that the ship resistances obtained from CFD simulations agree with the experimental measurements, especially with high values of the Froude number. The CFD simulations clearly show that a higher Froude number is associated with a higher resistance for all cases. The error between all the numerical and experimental results is less than 5-10%, except for the case with the lowest Froude number. With a Froude number of 0.813 (with a corresponding speed 1.716m/s), the numerical ship resistance is larger than 12% the measured ship resistance.

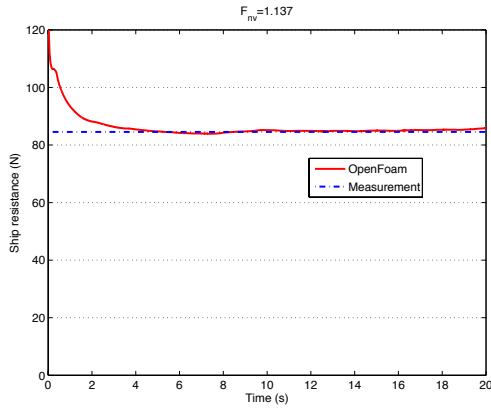


Figure 4(c): The ship resistance at $F_{nv} = 1.137$

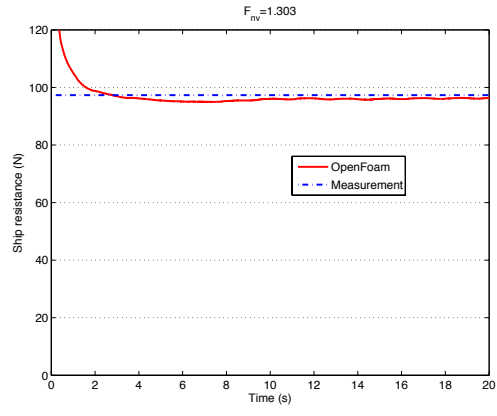


Figure 4(d): The ship resistance at $F_{nv} = 1.303$

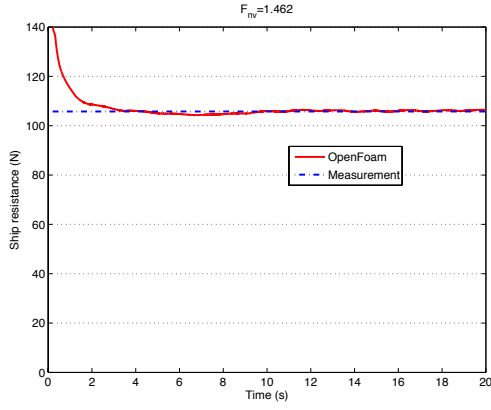


Figure 4(e): The ship resistance at $F_{nv} = 1.462$

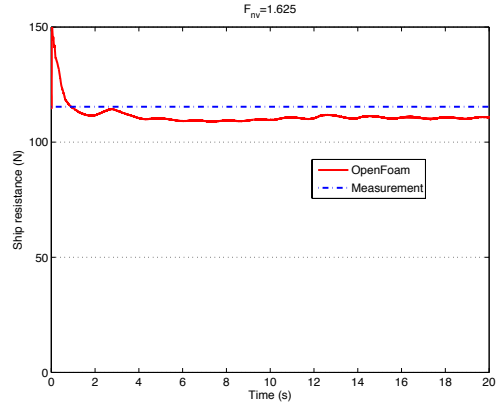


Figure 4(f): The ship resistance at $F_{nv} = 1.625$

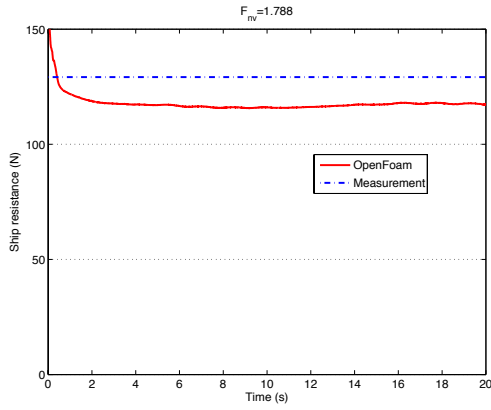


Figure 4(g): The ship resistance at $F_{nv} = 1.788$

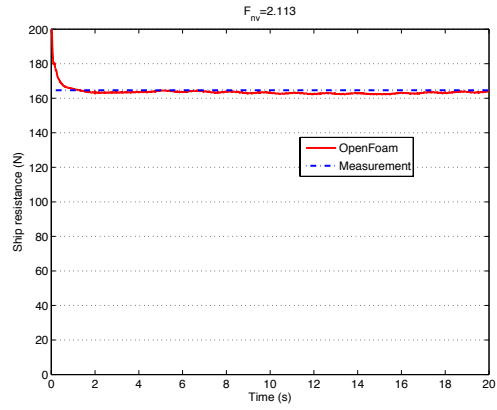


Figure 4(h): The ship resistance at $F_{nv} = 2.113$

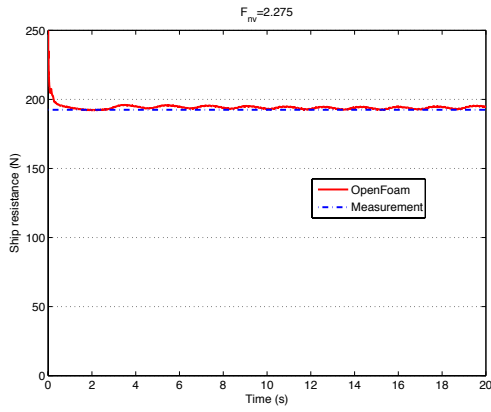


Figure 4(i): The ship resistance at $F_{nv} = 2.275$

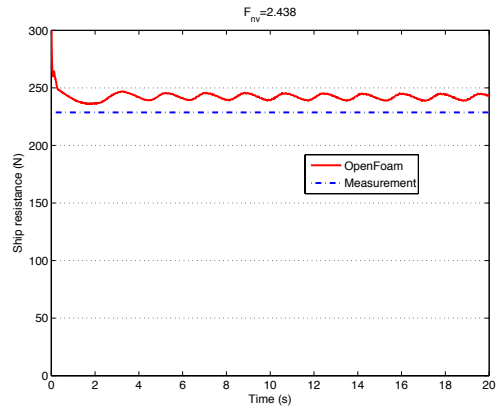


Figure 4(j): The ship resistance at $F_{nv} = 2.438$

4.2. Scalability

Since the hull is characterized by a certain geometrical complexity and its efficiency is directly influenced by the turbulent behavior produced by the given geometry, HPC methods are required in order to conduct mesh sensitivity analysis and to verify the modeling of the turbulence.

Speed-up tests have been carried out on Lindgren. The performance results for a mesh consisting of 1.4M points and 1.3M cells are shown in Table 2. Referring to Table 2, an additional speed-up vs. number of cores is shown in Figure 5. The speed-up is linear between 6 to 72 cores. For less than 72 cores, a super-linear behavior is observed when the partition tool Scotch is used. Moreover, an efficiency of 72.9% can be achieved for up to 144 cores.

#CPUs	Time (sec.)/step	Speed-up
6	23.5	1.0
12	11.7	2.0
24	5.6	4.2
48	2.8	8.5
60	2.3	10.3
72	2.0	11.7
96	1.6	14.9
120	1.4	16.5
144	1.3	17.5

Table 2. The time and speed-up for the application at Froude Number 0.975

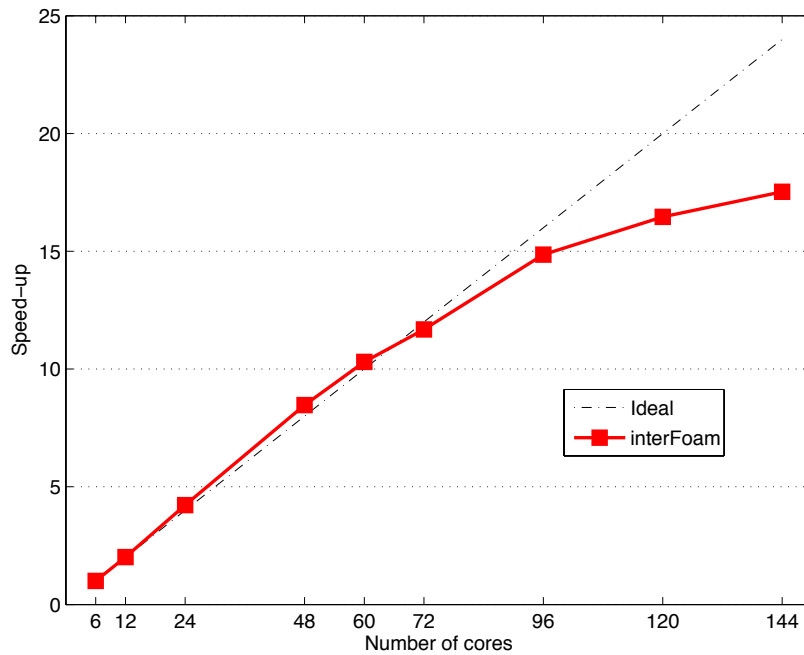


Figure 5

5. Conclusions

We have verified that OpenFOAM is an accurate and reliable CFD tool for MONOTRICAT hull resistance simulations. The numerical results agree well with the experimental measurements. Although some further investigation is needed to understand how better agreement can be obtained, the absolute and relative accuracy of the CFD predictions appear to be of the same order of magnitude as that expected from the real-world tests and thus these simulation methods can be used for optimization purposes for these type of vessels.

The scalability tests illustrate that a linear speed-up can be obtained using between 6 and 72 cores and that an efficiency of 72.9% can be achieved for up to 144 cores. The accurate numerical results and the scalability tests are very encouraging as they make it possible for Monotricat to focus on CFD simulations instead of basing their development process on traditional experiments.

Acknowledgements

This work was financially supported by the PRACE project funded in part by the EUs 7th Framework Programme (FP7/2007-2013) under grant agreement no. RI-312763. The work was done using the PRACE Research Infrastructure Tier-1 resource Lindgren at KTH-PDC, Sweden. We would like to thank Prof. Kevin Maki and Andreas Penza for constructive discussion in relation to this project.

References

- [1] Monotricat SRL, <http://monotricat.wix.com/monotricat>
- [2] OpenFOAM, <http://www.openfoam.org>
- [3] snappyHexMesh, <https://sites.google.com/site/snappywiki/snappyhexmesh>
- [4] K. Maki, Ship Resistance Simulations with OpenFOAM (Slides), 2011
- [5] L. Larsson, F. Stern, M. Vissonneau, CFD in ship hydrodynamics
- [6] S. Sadiq and Xiong-liang, Multi-Dimensional Numerical Free Surface VOF modeling with Moon Pool Experiments, in Proceedings of ASME 27th International Conference on Offshore Mechanics and Arctic Engineering, Estoril, 2008
- [7] Y. M. Ahmed, Numerical simulation for the free surface flow around a complex ship hull form at different Froude numbers
- [8] L. Larsson and H.C.Raven, The Principles of Naval Architecture Series; Ship resistance and flow, The Society of Naval Architects and Marine Engineers, 2010
- [9] O. M. Faltinsen, Hydrodynamics of High-Speed Marine Vehicles, Cambridge University Press, 2005
- [10] F.R. Menter, Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows, AIAA Paper 93-2906
- [11] F.R. Menter Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, vol. 32, no 8. pp. 1598-1605.
- [12] D. C. Wilcox, Turbulence Modelling for CFD, 1998
- [13] SST K-omega model http://www.cfd-online.com/Wiki/SST_k-omega_model
- [14] A. Penza, Hydrodynamics application for standard DTMB 5415 bare hull with OpenFOAM (slides)