This is an electronic reprint of the original article.
This reprint may differ from the original in pagination and typographic detail.

Peltonen, Petteri; Kanninen, Pekka; Laurila, Erkki; Vuorinen, Ville

The ghost fluid method for OpenFOAM

Published in:
Ocean Engineering

DOI:
[10.1016/j.oceaneng.2020.108007](https://doi.org/10.1016/j.oceaneng.2020.108007)

Published: 15/11/2020

Document Version
Peer reviewed version

Published under the following license:
CC BY-NC-ND

Please cite the original version:
Peltonen, P., Kanninen, P., Laurila, E., & Vuorinen, V. (2020). The ghost fluid method for OpenFOAM: A comparative study in marine context. *Ocean Engineering*, 216, [108007].
<https://doi.org/10.1016/j.oceaneng.2020.108007>

1 The ghost fluid method for OpenFOAM: a comparative 2 study in marine context

3 Petteri Peltonen^{a,*}, Pekka Kanninen^b, Erkki Laurila^a, Ville Vuorinen^a

4 ^a*Aalto University, Department of Mechanical Engineering, FI-00076 Aalto, Finland*

5 ^b*Meyer Turku, P.O. Box 666, Telakkakatu 1, FI-20101 Turku, Finland*

6 Abstract

7 The open source CFD library OpenFOAM is utilized to implement the
8 ghost fluid method (GFM). The implemented solver **GFMFoam** is compared
9 against the standard volume-of-fluid (VOF) solver **interFoam** in OpenFOAM.
10 For both VOF solvers, the scalar transport equation is solved similarly but
11 the discretization of the momentum and continuity equations differ between
12 **GFMFoam** and **interFoam**. The two methods are compared in four test cases
13 with mild surface deformation. First, the GFM implementation is validated.
14 Second, the results on a 2d interface shear layer highlight the importance
15 of the viscous term treatment in the GFM. Third, the model scale Wiggley
16 hull results indicate that the **GFMFoam** is more sensitive to the grid aspect
17 ratio and resolution than **interFoam** but the viscous resistance is better pre-
18 dicted. Also, **GFMFoam** is shown to be less sensitive to numerical ventilation
19 than **interFoam**. Fourth, the results from the Hamburg Test Case indicate a
20 total resistance prediction within 1% when evaluated against reference data
21 at model scale and 10% at full scale. Based on the steady state test cases
22 of the present study, **GFMFoam** has moderate benefits over **interFoam** but it
23 does not clearly outperform **interFoam**.

24 *Keywords:* CFD, Free surface flow, Ghost fluid method, VOF, Marine

25 1. Introduction

26 Computational fluid dynamics (CFD) has an essential role in the research
27 and development phase of ships when aiming towards resistance reduction.
28 In fact, predictive CFD approaches, with accuracy comparable to real world
29 towing tests, have been developed [1–4]. Thereby, tests and CFD simula-
30 tions supplement one another on the model scale, yet, full scale comparison

31 is typically unpractical in marine context. The shift from model scale simu-
32 lations to full scale simulations is not straightforward due to the increase in
33 the Reynolds number. The increased Reynolds number in full scale leads to
34 a higher resolution requirement in the boundary layer which, as a practical
35 trade-off, may result in a coarser grid outside the boundary layer (e.g. at the
36 free surface).

37 An accurate ship resistance simulation requires the solution of both the
38 gas and the liquid phase as well as the position of the interface between the
39 two phases. An illustration of the wave pattern for a typical hull simulation
40 is presented in Figure 1. On a discrete grid, the air-water interface is discon-
41 tinuous. In fact, discretization of the governing equations in the vicinity of
42 the discontinuity is of high priority in practice. Over the past years, several
43 methods have been proposed for multiphase flow simulations and especially
44 for the treatment of the interface [5].

45 Perhaps the most common two-phase flow simulation approach is the
46 volume-of-fluid (VOF) method, first introduced by Hirt [6]. The idea in VOF
47 is to transport a marker field describing either air or water and to compute
48 the fluid properties (density and viscosity) using the marker field. Advec-
49 tion of the marker field such that the interface remains sharp is non-trivial
50 and many different methods have been developed. The interface capturing
51 approaches can be divided into two groups: geometric and algebraic. In
52 geometric formulations, an explicit (sub-cell) reconstruction of the interface
53 is used in computing the volume fluxes. In algebraic formulations, no geo-
54 metrical reconstruction is required. Instead, the discrete form of the marker
55 field equation is formulated such that the interface smearing is minimized.
56 Examples of the geometric formulations are, for example, the piecewise lin-
57 ear interface calculation (PLIC) [7] and the isoAdvector method [8]. Popular
58 algebraic formulations include tangent of hyperbola interface capturing with
59 slope weighting (THINC) [9] and the multidimensional universal limiter with
60 explicit solution (MULES) [10].

Figure 1: Illustration of the wave pattern for a typical hull simulation (top) and the velocity field obtained with a traditional VOF method (bottom left) and with GFM (bottom right) for a bubble convection in zero gravity. The flow direction is from left to right in the bottom two pictures.

61 Transporting the scalar field accurately may not be sufficient for overall
62 numerical accuracy. Rather recently, it has been proposed that the coupling
63 between the momentum and scalar equation is also important in VOF formu-
64 lations [11]. If the momentum and continuity equations are solved without

suitable interface discretization, spurious velocities may occur at the vicinity of the interface (see Figure 1). Vukčević et al. [11] state that these spurious velocities are due to the imbalance of the dynamic pressure gradient and the density gradient which occurs even for hydrostatic simulations when a segregated (pressure correction) solution algorithm is used. The unphysical spurious velocities can be problematic as the resulting velocity gradients at the interface may influence the turbulence production in the underlying turbulence models. This issue is more common in full scale ship simulations where the Reynolds number is higher and the relative grid resolution lower than in model scale simulations. The ghost fluid method (GFM) has been proposed as an approach to overcome certain issues of the traditional VOF method. In the context of the present paper, 'traditional VOF' refers to a VOF method without special interface treatment in the momentum and pressure equations.

The GFM was originally developed by Fedkiw et al. [12] for multiphase Euler equations solved on a Cartesian grid. Later Kang et al. extended the method for viscous incompressible flows [13]. In both papers, the level set method was used to describe the interface. The first application of the GFM on an unstructured grid was presented by Shin [14]. In the paper, incompressible and inviscid flow over a submerged hydrofoil was studied and the obtained results showed good agreement with the corresponding experiments.

In marine context, GFM was first applied by Huang et al. [15]. The authors utilized the GFM with the level-set method for a two-dimensional submerged bump and a DTMB-5415 combatant hull in model scale. For both cases, comparison with experiments and single-phase level-set simulations was presented. When compared to the single-phase simulations, the GFM results showed a sharper interface between air and water because no artificial thickening (smoothing) of the interface is required with the GFM. In the submerged bump test case, the authors stated that the thickening of the interface results in unphysical velocity field on the air side of the single-phase level-set results. In the hull simulations using the GFM approach, the authors stated that the obtained velocity field is continuous across the interface and no spurious velocities are generated.

In fact, the ability of the GFM to retain a continuous velocity field across the interface has been observed by others as well. Queutey & Visonneau [16] implemented the GFM in the finite volume VOF context for an arbitrary unstructured grid. The test cases presented by the authors included the Rayleigh-Taylor instability and a flow over a Series 60 hull on the model

103 scale. It was observed that the velocity field obtained with the GFM is
 104 continuous but requires a sufficient grid density at the interface. The authors
 105 also stated that finding a grid independent solution may be challenging due
 106 to the significant changes observed for the finest studied grid and the second
 107 finest grid.

108 Following the work by Huang et al., Vukčević et al. [11] have provided
 109 a detailed description of the implementation of the GFM in the computa-
 110 tional fluid dynamics (CFD) library OpenFOAM. In fact, the description
 111 by Vukčević et al. was thorough enough to serve as a basis for the imple-
 112 mentation in the present paper. In addition to the description of the solver,
 113 Vukčević et al. presented results for a two-dimensional ramp and a Kriso
 114 container ship with good correspondence to the reference data. The authors
 115 did not provide comparison with the `interFoam` solver.

116 In this paper, we present a comparison of two volume-of-fluid [6] (VOF)
 117 methods in marine context. The first method is the ghost-fluid method
 118 (GFM) [12], where special treatment is used for the discrete operators near
 119 the interface. Hereafter, the first method is referred to as `GFMFoam`. The
 120 second method is a traditional VOF method where the discontinuity at the
 121 air-water interface is discretized using conventional discretization methods.
 122 Hereafter, the second method is referred to as `interFoam` [17]. We note that
 123 `interFoam` is the standard VOF solver available in the OpenFOAM-v1806
 124 CFD-library [18] and the `GFMFoam` is implemented to the same library by the
 125 authors. The implementation of the `GFMFoam` is based on the description by
 126 Vukčević et al. [11]. Since the focus in the paper is on marine simulations,
 127 throughout this work, surface tension effects are neglected.

128 In marine context, `interFoam` has been successfully applied by many au-
 129 thors. Yang et al. [19] presented a thorough validation of the solver for five
 130 test cases including the Wigley hull with satisfactory results. Zha et al. [20]
 131 applied a modified version of `interFoam` to compute the resistance of a high
 132 speed catamaran and obtained good agreement with the corresponding ex-
 133 periments. Park et al. [21] studied the effects of the near-wall grid resolution
 134 on the total resistance of the KRISO container ship using `interFoam`. The
 135 authors noted a sensitivity of the wall shear stress to the wall-normal grid
 136 resolution.

137 Based on the previous work, it can be concluded that the GFM retains
 138 a continuous velocity field across the interface. Compared to methods which
 139 generate spurious velocities at the interface, the continuous velocity field
 140 may enable for: taking larger time steps, more accurate solution of the VOF

141 marker equation and better turbulence model behaviour at the interface.
 142 However, also the `interFoam` solver has been successfully used in the past.
 143 Thereby, based on the literature, we consider that the potential benefits of
 144 GFM over traditional VOF approaches are still unclear. In the present work
 145 we aim to identify the benefits and drawbacks of the GFM compared to
 146 the traditional interface discretization of `interFoam` in marine context. The
 147 objectives of the present work are formulated as follows:

- 148 • Assess the key aspects and potential caveats of implementing GFM in
 149 OpenFOAM.
- 150 • Compare `GFMFoam` with `interFoam` for benchmark cases with mild sur-
 151 face deformation and quantify the benefits and drawbacks.

152 2. Methods

153 In the present study, two solvers are compared for two-phase flow: `GFMFoam`
 154 and `interFoam`. Both solvers employ the VOF method in the finite volume
 155 context. We note that in both solvers the VOF marker field is transported
 156 with the MULES [10] approach but coupling of the marker field to the con-
 157 tinuity and momentum equations is different between the solvers. Here, a
 158 brief description of the studied solvers is presented.

159 2.1. *interFoam*

The continuity and momentum equations (as solved in `interFoam`) for two-phase, incompressible and immiscible fluids with negligible surface tension are given as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (1)$$

$$\frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) - \nabla \cdot (\mu (\nabla \vec{u} + \nabla \vec{u}^T)) = -\nabla p_d - \vec{g} \cdot \vec{x} \nabla \rho, \quad (2)$$

where $\vec{u}$, ρ , p_d and μ are the velocity, density, dynamic pressure ($p_d = p - \rho \vec{g} \cdot \vec{x}$) and dynamic viscosity, respectively. In addition to the continuity and momentum equations a scalar transport equation for the marker field α is solved using the MULES approach

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{u}) + \nabla \cdot (\vec{u}_c \alpha (1 - \alpha)) = 0. \quad (3)$$

160 The last term on the LHS is as an artificial compression term which is added
 161 to avoid smearing of the interface and the term is only active in the vicinity
 162 of the interface [10, 17].

The main parts of the **interFoam** solution algorithm are summarized in Figure 2 for a single time step. The algorithm is a combination of PISO (inner) and SIMPLE (outer) iterations. The scalar transport and momentum equations are solved only in the outer loop and pressure is corrected within the inner loop. As can be seen from Figure 2, the algorithm begins with a solution of the marker field after which the viscosity, density and mass flux ($\dot{m}$) are updated from the volume fraction

$$\begin{aligned}\rho &= \rho_L \alpha + \rho_G (1 - \alpha) \\ \mu &= \mu_L \alpha + \mu_G (1 - \alpha),\end{aligned}\tag{4}$$

163 where the subscripts L and G denote liquid and gas, respectively.

164 The next step is the assembly of the discrete momentum equation. When
 165 the momentum equation is assembled the (implicit) LHS terms are always
 166 considered and the (explicit) RHS terms for the pressure and density gradi-
 167 ents are added as a source term only if the user chooses to solve the momen-
 168 tum equation at this point. The optional solution of the momentum equation
 169 at this point is referred to as the momentum predictor. We note that the
 170 solution of the momentum predictor at this point is not necessary and it is
 171 not solved in the present work because of the steady state assumption in all
 172 simulations. The algorithm proceeds with the assembly and solution of the
 173 pressure Poisson equation followed by a correction of the velocity. For more
 174 details on the **interFoam** implementation the reader is referred to [17].

Figure 2: Main parts of the VOF solver algorithm within a single time step.

175 2.2. *GFMFoam*

In *GFMFoam*, we assume that for both phases separately the incompressible N-S formulation applies:

$$\nabla \cdot \vec{u} = 0 \quad (5)$$

$$\frac{\partial \vec{u}}{\partial t} + \nabla \cdot (\vec{u}\vec{u}) - \beta \nabla \cdot (\mu(\nabla \vec{u} + \nabla \vec{u}^T)) = -\beta \nabla p_d, \quad (6)$$

176 where β is the inverse density either $1/\rho_L$ for the liquid or $1/\rho_G$ for the gas.
 177 Hereafter, we will omit the subscript d from the dynamic pressure. Here β is
 178 left outside the divergence in the viscous term on purpose and a remark on
 179 the viscous treatment in *GFMFoam* is given in Section 2.2.1.

180 The idea in the GFM is to build the discrete operators in the NS equations
 181 using one-sided differences on each side of the interface taking into account
 182 the jump conditions [15]. As the solution algorithm presented in Figure 2 is

183 the same for both `interFoam` and `GFMFoam`, here, we present a short descrip-
 184 tion on the interface corrected interpolation. For more details, the reader is
 185 referred to the paper by Vukčević et al. [11].

186 Figure 3 shows a schematic of a linear interpolation (Figure 3a) and a
 187 jump-corrected interpolation (Figure 3b) for obtaining the cell face value
 188 p_f from the cell center values p_P and p_N . To evaluate the face value in
 189 the GFM, one-sided extrapolants to an imaginary "ghost" neighbour are
 190 constructed. For cell P the value p_P^- is extrapolated and for cell N the value
 191 p_P^+ is extrapolated. The idea in the extrapolation is to first generate $p^\pm$
 192 values infinitesimally close to the free surface on each side and to linearly
 193 extrapolate through the cell value (p_P or p_N) and the interface value (either
 194 p^+ or p^-). An example on how the interface corrected interpolation can be
 195 used to compute the pressure gradient is given in Appendix A.

Figure 3: Interpolation examples. Subscripts P and N denote cell center values and f denotes the cell face. The interface is at location Γ_f .

To determine when to use the interface corrected discretization we require a condition for determining if a cell pair contains an interface in between. Following Vukčević et al. [11], a cell pair contains an interface if

$$(\alpha_P - 0.5)(\alpha_N - 0.5) < 0, \quad (7)$$

196 where α_P and α_N are volume fractions in cells P and N respectively. Same
 197 as in Vukčević et al. [11] we assume that the discrete interface location is
 198 located on a line between adjacent cell centers $x_\Gamma = \vec{x}_P + \lambda \vec{d}$, where $\vec{d}$ is a
 199 vector from cell center P to N and $\lambda = (\alpha_P - 0.5)/(\alpha_P - \alpha_N)$.

200 *2.2.1. Viscous term*

201 As mentioned before, the treatment of the viscous term in Eq. 6 differs
 202 slightly from that of Vukčević et al. [11]. Here, we present few possibilities
 203 for the treatment of the viscous term in the GFM.

The viscous term in the momentum equation of **GFMFoam** reads

$$\beta (\nabla \cdot (\mu(\nabla \vec{u} + \nabla \vec{u}^T))) = \beta (\nabla \cdot \mu \nabla \vec{u} + (\nabla \vec{u})^T \cdot \nabla \mu), \quad (8)$$

204 where μ is the mixture averaged viscosity from Eq. 4. In the discretization of
 205 the viscous term few possibilities exist. Either the GFM jump treatment [13]
 206 or discretization methods without jump treatment can be used for the viscous
 207 term. In the present paper, a combination of the two approaches is used. The
 208 first term on the RHS (with $\nabla \vec{u}$) is always discretized with a linear scheme
 209 and without jump correction at the interface. The second term is either
 210 neglected (**GFMFoam-V1**) or the gradient of μ is constructed in a manner
 211 analogous to the pressure gradient (**GFMFoam**) shown in Appendix A. That
 212 is, in **GFMFoam** an interface corrected discretization is used for the gradient of
 213 μ . The main difference to the paper by Vukčević et al. is that in the present
 214 paper, β is not moved inside the divergence. This leads to the interpolation of
 215 μ to the cell faces instead of ν and the discontinuous β field is used as a scaling
 216 factor. The benefit of the present approach is that with finite resolution at
 217 the interface, the water side shear stress is better captured as will be shown
 218 in Section 3.2. A summary of the different viscous treatments used in the
 219 present study is given in Table 1. In the table **GFMFoam-V2** corresponds to
 220 the treatment used by Vukčević et al.

Table 1: Summary of the different viscous treatments used in the present study.

ID	Viscous term	Momentum term	ϕ
GFMFoam	$\beta(\nabla \cdot (\mu \nabla \vec{u}) + (\nabla \vec{u})^T \cdot \nabla \mu)$	$\nabla \cdot (\phi \vec{u})$	$\vec{u} \cdot \vec{n}$
GFMFoam-V1	$\beta(\nabla \cdot (\mu \nabla \vec{u}))$	$\nabla \cdot (\phi \vec{u})$	$\vec{u} \cdot \vec{n}$
GFMFoam-V2	$\nabla \cdot (\nu \nabla \vec{u})$	$\nabla \cdot (\phi \vec{u})$	$\vec{u} \cdot \vec{n}$
interFoam	$\nabla \cdot (\mu \nabla \vec{u})$	$\nabla \cdot (\phi \vec{u})$	$\rho \vec{u} \cdot \vec{n}$

221 *2.3. Discretization schemes and the applied turbulence model*

222 Due to the steady-state assumption in all test cases, the time deriva-
 223 tives are computed with a local time stepping scheme (**localEuler** in Open-
 224 FOAM) to accelerate convergence. The **localEuler** time scheme is a first

225 order accurate pseudo-transient scheme enabling the use of a spatially varying
 226 time step to accelerate convergence to steady-state. The input parameters
 227 for the `localEuler` scheme are presented in Appendix B. Convective terms in
 228 the momentum equation are solved with the flux-limited `Gauss vanAlbada`
 229 scheme and the convective terms in the VOF scalar transport equation are
 230 solved with the flux-limited `Gauss vanLeer` scheme. Linear schemes are
 231 used for all other terms (with interface correction in the `GFMFoam`). In the
 232 two-dimensional laminar test cases no turbulence model is used. In the three-
 233 dimensional test cases (Wigley hull and HTC) Menter’s SST $k - \omega$ model is
 234 used and the boundary layers are modelled with wall functions.

235 The air-water interface is defined as the $\alpha = 0.5$ contour constructed with
 236 second order accuracy from the discrete cell values using the post processing
 237 functions in OpenFOAM.

238 3. Results

239 3.1. Two-dimensional inviscid flow over a step

240 The first studied test case is a two-dimensional inviscid flow over a step for
 241 which an analytic solution for the outlet water height h_0 exists (see e.g. [22]).
 242 The same test case was used by Vukčević et al. [11] in their implementation
 243 of the GFM. The domain and the grid topology are presented in Figure 4.
 244 The geometry consists of an inlet boundary (on the left) where the water
 245 height is fixed to h and an outlet boundary (on the right) where the water
 246 height h_o is an output of the simulation. The top and the bottom boundary
 247 are treated as slip walls and the step height on the bottom boundary is $h/5$.
 248 At the inlet boundary the Dirichlet condition is applied for $\vec{u}$ and α while
 249 the Neumann condition (zero gradient) is applied for the dynamic pressure.
 250 On the outlet boundary the Dirichlet condition is applied for the dynamic
 251 pressure and the Neumann condition is set for $\vec{u}$ and α . The simulations are
 252 run at $Fr = |\vec{u}| \sqrt{|\vec{g}| h} = 1.92$ and a density ratio of 1000.

253 The test case is studied at 7 different resolutions: $n = 5, 10, 20, 40, 80, 160, 320$
 254 cells per inlet water height. The grid size in both spatial directions is the
 255 same. The outlet water height and the relative difference to the water height
 256 computed with the densest grid are presented in Figure 5. As can be seen,
 257 both `interFoam` and `GFMFoam` eventually converge to the same $h_o \approx 1.11$ as
 258 the grid is refined. This value is different than the analytic solution based on
 259 the Bernoulli equation $h_o = 1.08973$. For this reason, the grid convergence
 260 results are presented relative to the outlet water height obtained with the

261 densest grid. The trend in the grid convergence is similar with both solvers.
 262 Both solvers indicate a rate of convergence between $\mathcal{O}(n)$ and $\mathcal{O}(n^2)$ which is
 263 to be expected. We note that we observed spurious velocity in the `interFoam`
 264 results at the interface, however, the effect of the spurious velocity was noted
 265 to be minor on the outlet water height. The reason the for the minor effect
 266 may be due to the inviscid flow assumption in this test case.

Figure 4: The geometry of the domain in the two-dimensional step test case. The grid lines shown in the figure correspond to the coarsest mesh resolution. Flow direction is from left to right.

Figure 5: Water height as a function of cell count per initial water height (left) and the relative difference of the outlet water height to the outlet water height computed with the densest $n/h = 320$ grid (right). The dashed line corresponds to the analytic solution [22] based on the Bernoulli equation.

267 3.2. Two-dimensional interface shear layer

268 In the previous test case the viscous effects were neglected. To point out
 269 the differences in the viscous term treatment we consider two-dimensional
 270 shearing of air and water, where the treatment of the viscous term is of high
 271 importance. The case consists of a still layer of water with a moving air layer
 272 above inducing a growing shear layer, mostly on the air side. A similarity
 273 solution for the interface velocity and shear stress profile was derived by
 274 Lock [23].

275 A sketch of the case set-up is presented in Figure 6. The rectangular
 276 domain has two inlets (air and water) which are split initially at $x < 0$
 277 by a zero thickness wall (baffle). The baffle is used to minimize the large
 278 stagnation pressure gradient at the $x = y = 0$ convecting downstream. The
 279 case is run with material properties of air and water at a temperature of
 280 10°C ($k = \rho_2 \mu_2 / \rho_1 \mu_1 = 5.965 \times 10^4$) and for air inflow velocity $U = 0.1 \text{ m/s}$.
 281 The top and bottom boundaries are set as symmetry planes and the inlet
 282 and outlet conditions are identical to those in the two-dimensional step test
 283 case. The test case is studied with grid resolutions $N_x \times N_y = 80 \times 30$ and
 284 120×120 . The cells are graded in y direction towards the interface such that
 285 the grid size δ_y at $y = 0$ is 40 times smaller than at the freestream $y = \pm H/2$.

Figure 6: The case set-up in the shear interface shear layer case. The dimensions of the domain are $H = 3 \text{ m}$ and $L = 6.25 \text{ m}$ and the length of the zero thickness slip wall (baffle) is 1.25 m .

Figure 7 shows the simulated non-dimensional velocity and shear stress profiles through the interface along with the similarity solution [23]. The simulation profiles are averaged from three x -coordinate locations to ensure that the growth of the shear layer is properly captured. The dimensionless y -coordinate is $\eta_i = y(\frac{U}{x\nu_i})^{1/2}$ and the shear stress $f_i'' = \frac{\partial u}{\partial x}/(U(\frac{U}{\nu_i x})^{1/2})$ with i denoting air or water. As can be seen from Figure 7a **interFoam** shows a thickening of the shear layer and a clear overshoot of velocity with the coarser grid. The overshoot is caused by the spurious velocity occurring in the **interFoam** results due to the imbalance of the pressure and density gradients in the pressure correction as shown by Vukčević et al. [11]. In the **interFoam** results, we noticed that the magnitude of the spurious velocity increased as the grid was coarsened or the aspect ratio (δ_x/δ_y) of the cells was increased. When the grid is refined (Figure 7b) the velocity profile obtained with **GFMFoam**, **GFMFoam-V1** and **interFoam** are fairly close to the reference similarity solution. It should be noted that with the dense grid, there are approximately ten cells in the shear layer $0 \leq \eta \leq 6$. Such resolution is not feasible in a hull simulation, for example.

The most important observation to be made from Figure 7 is the deviation of **GFMFoam-V2** result from the reference solution and other simulations. The deviation does not decrease when the grid is refined. The reason for this can be understood in a straightforward manner. In the momentum equation (Eq. 6) of the GFM solvers, the momentum flux is defined without density, therefore, it is necessary to use the dynamic viscosity in the viscous term. If the kinematic viscosity is used instead (as in **GFMFoam-V2**), the pressure gradient is the only term containing the density. Without additional jump treatment for the viscous term, the lighter fluid will cause a long reaching shear layer on the heavy fluid side although the density difference is large. Such shear layer is observed in the **GFMFoam-V2** result presented in Figure 7.

The difference in the viscous treatments of **GFMFoam** and **GFMFoam-V1** is the additional term $(\nabla \vec{u})^T \cdot \nabla \mu$. The effect of this term is better seen from the shear stress profiles in Figure 7b. **GFMFoam-V1** does not include this term and the results show a moderate spike in the shear stress in the vicinity of the interface.

(a) 80×30 grid.

(b) 120×120 grid.

Figure 7: Dimensionless velocity (u/U) and shear stress (f'') through the interface. The simulated results are an average of three profiles taken at locations $x = 0.5, 0.75$ and 1 m and the dots are the similarity solution by Lock [23].

319 3.3. Wigley hull

320 3.3.1. Wigley hull case set-up

321 The third test case is the parabolic Wigley hull profile. The hull shape
 322 is $y = B(1 - (\frac{z}{T})^2)(1 - (2\frac{x}{L})^2)$ for beam B , draft T and length L . The
 323 dimensions of the ship used in the present study are set to correspond the
 324 experiments by Kajitani et al. [24]. Only half of the hull is modelled due to
 325 the symmetry. The grid topology and domain dimensions are presented in
 326 Figure 8. The rest of the input parameters are presented in Table 2.

Figure 8: The mesh topology (every third grid line of the coarsest mesh shown) and domain dimensions of the Wigley hull test case.

Table 2: The input parameters in the Wigley hull test case. The Reynolds number is referred to the hull length and viscosity of water $Re = UL/\nu_L$.

Dimensions [m]			Fr		$Re \times 10^6$		ρ [kg/m ³]		$\nu \times 10^6$ [m ² /s]	
L	B	T	min	max	min	max	air	water	air	water
2.5	0.25	0.156	0.25	0.4	1.94	5.17	997.6	1.201	15.2	95.8

At the inlet of the domain velocity is fixed and the Neumann condition is applied for the pressure. The inlet boundary is split into two sections from $z = 0$ such that $z > 0$ is the air region where $\alpha = 0$ is applied and $z < 0$ where $\alpha = 1$ is applied. At the outlet boundary, the Neumann condition is applied for both U and α whereas pressure is fixed. On the hull surface the no-slip condition is used. All other boundaries are treated as symmetry planes.

The simulations are carried out with three different mesh resolutions and for 25 linearly spaced Froude numbers $Fr = U/\sqrt{gL} \in [0.15, 0.4]$. In order to limit the total number of simulations and due to the close correspondence between `GFMFoam`, `GFMFoam-V1` and `interFoam`, the comparison is carried out only for `GFMFoam` and `interFoam`. Altogether, this corresponds to 150 simulations for `GFMFoam` (75) and `interFoam` (75). The meshes are generated such that the boundary layer grid size (δ_y) on the hull surface is kept constant for all grids and the refinement occurs without topology change outside the boundary layer. For the smallest $Fr = 0.15$ the first cell size in wall units is $\delta_y^+ = \delta_y u_\tau / \nu \approx 37$ and for the largest $Fr = 0.4$ the cell size in wall units is $\delta_y^+ \approx 87$. The difference is due to the increase of the friction velocity $u_\tau = \sqrt{\tau_w / \rho}$ when the Fr is increased. The near-wall resolution is typical in model scale marine applications and it is consistent with the wall-model assumptions in the present RANS simulations. Some details of the studied meshes are presented in Table 3 and the script used to generate the mesh in OpenFOAM format is provided online [25].

Table 3: The studied mesh resolutions in the Wigley hull test case. The dimensionless grid spacings ($\delta_x^+, \delta_y^+, \delta_z^+$) are in wall units and correspond to the maximum values on the hull surface.

	$N_x \times N_y \times N_z$	N_{total}	$\delta_x^+, \delta_y^+, \delta_z^+$	
			$Fr = 0.15$	$Fr = 0.4$
Mesh 1	$100 \times 66 \times 42$	277×10^3	1250, 36, 208	2925, 84, 487
Mesh 2	$180 \times 110 \times 70$	1.4×10^6	784, 38, 131	1831, 88, 305
Mesh 3	$270 \times 165 \times 105$	4.7×10^6	532, 38, 89	1240, 89, 206

3.3.2. Wigley hull results

The total resistance coefficients from the Wigley hull simulations are shown in Figure 9. As can be seen from the **GFMFoam** results, the correspondence between the simulations and experiments for the total resistance at the coarsest resolution (Mesh 1) is poor. Such observation is likely due to the large aspect ratio for the cells in wall normal direction for Mesh 1 because of the chosen refinement strategy (constant δ_y^+ for the boundary layer cell height). On the other hand, **interFoam** results are still reasonable for Mesh 1. For the two denser resolutions, the total resistance is well predicted until $Fr \approx 0.35$ by both solvers.

With Mesh 2 and Mesh 3 the total resistance with **interFoam** deviates from the experiments slightly more than **GFMFoam** for $Fr < 0.35$. However, the friction resistance characteristics for **interFoam** is noted to be peculiar in Figure 9. With $Fr > 0.3$, the friction resistance from the **interFoam** simulations starts to deviate from the ITTC-57 friction line correlation. Counter-intuitively, the deviation tends to increase with increasing grid resolution. The reason for the deviation is revealed in Figure 10, which shows the norm of the wall shear stress ($\tau_w = (\nu + \nu_t)\nabla\vec{u} \cdot \vec{n}$) and the modelled turbulence viscosity (ν_t) at the surface of the hull. At the midship and close to the waterline, **interFoam** shows higher values of τ_w than **GFMFoam**. The higher τ_w values are a direct consequence of the high ν_t in the respective location. Figure 10 also shows numerical ventilation with the **interFoam**, which is not observed in the **GFMFoam** results. The numerical ventilation was related to the aspect ratio of the cells in the vicinity of the hull with Mesh 1 showing the largest amount of ventilation in **interFoam** results.

Figures 11 and 12 show cutplanes from the midship ($y - z$ plane) of the velocity, velocity gradient and turbulent kinetic energy fields. At $Fr = 0.253$ (Figure 11) the fields are quite similar between the two solvers and for all

378 meshes. At $Fr = 0.305$ (Figure 12) **interFoam** shows a stripe of high velocity
379 and velocity gradient at the free surface. The velocity gradient, however, does
380 not reach on to the hull surface and the reason for this is the increase of the
381 turbulence kinetic energy next to the hull which tends to smear the gradient.
382 We therefore conclude that the high friction resistance for **interFoam** at high
383 Fr and increasing grid resolution is a result of the high turbulence production
384 at the interface. It is not, however, clear that the observed spurious velocity
385 caused by the interface discretization is solely the cause of the high turbulence
386 production. Previously [26], it has been noticed that the underlying RANS
387 model suffers from overproduction of turbulence for two phase flows. But as
388 the high production is only observed in the present **interFoam** results it is
389 likely that the observed spurious velocity has an impact on the production
390 of turbulence at the interface.

Figure 9: The total C_t , pressure C_p and friction force C_μ coefficients from the Wigley hull simulations from top to bottom, respectively. `interFoam` results are on the left column and `GFMFoam` on the right column. The markers are shown for every second simulated Fr . The reference line (red) in C_t plot corresponds to the total resistance measurements by Kajitani et al. [24] and in the C_μ plot the reference line is the ITTC-57 friction line correlation ($C_\mu = 0.075 / (\log_{10} Re - 2)^2$). The C_p reference is the difference $C_p = C_t - C_\mu$.

Figure 10: Wall shear stress, turbulence viscosity and the marker scalar field at the surface of the hull. The results are taken at $Fr = 0.305$ with Mesh 3 and bow is on the left in the figure.

391 Figures 11 and 12 show midship cut planes of velocity, velocity gradient
 392 and turbulence kinetic for $Fr = 0.253$ and $Fr = 0.305$, respectively.

Figure 11: Slices from the $y-z$ plane at $x = 0$ (mid ship) at $Fr = 0.253$. In each subfigure the grid resolution varies from Mesh 1 to Mesh 3 from top to bottom, respectively.

Figure 12: Slices from the $y-z$ plane at $x = 0$ (mid ship) at $Fr = 0.305$. In each subfigure the grid resolution varies from Mesh 1 to Mesh 3 from top to bottom, respectively.

Figure 13 shows the wave patterns for all studied meshes at $Fr = 0.305$. Quite intuitively, a longer extending wave pattern is observed with increasing resolution for both solvers. The largest differences in the wave shapes between the two solvers can be observed from the wave bending at a certain distance from the hull indicated by the arrows in the figure. For `interFoam`

398 the waves are bent in the bow direction and for **GFMFoam** in the stern (stream-
 399 wise) direction. The reason for the peculiar wave shape with **GFMFoam** is a
 400 combination of two factors: 1) the definition used to mark the interface
 401 cell faces for the GFM treatment and 2) decreasing grid resolution further
 402 away from the hull. The GFM discretization procedure is used for cell pairs
 403 $\alpha_P > 0.5$ and $\alpha_N < 0.5$ or vice versa. This means that the face between a cell
 404 pair containing $\alpha_P = 0.49$ and $\alpha_N = 0.01$ is not treated as an interface. Such
 405 cell pairs are frequently observed in regions where the wave height is small
 406 and resolution in the directions (y and x in Figure 13) not normal to the
 407 free surface is large. In other words, if the well-known dam break problem
 408 was studied with **GFMFoam** for initial values $\alpha_1 = 0.49$ and $\alpha_2 = 0$, the initial
 409 condition would be retained throughout the simulation. This effect is further
 410 illustrated in Appendix C.

Figure 13: The wave elevation in the Wigley hull case for the three studied resolutions at $Fr = 0.305$. The 20 contour lines shown are linearly spaced in the range $z/T \in [-0.15, 0.15]$.

411 3.4. The Hamburg test case

412 3.4.1. HTC set-up

413 The final test case of the present study is the Hamburg Test Case (HTC) [27].
 414 The purpose of the HTC simulations is to study the free surface phenom-
 415 ena in an engineering context and to relate the previous observations on the
 416 Wiggley hull with a real vessel. Here, the Froude number is $Fr = 0.238$ and
 417 the RANS simulations are carried out both on the model ($Re = 1.177 \times 10^7$)
 418 and full scale ($Re = 1.2 \times 10^9$). The physical dimensions of the ship and
 419 details of the grids used are presented in Table 4.

420 The size of the rectangular computational domain is $5 \times 1.5 \times 2.3 L_{pp}$ in
 421 the streamwise, beam and draught directions, respectively. Only a half-hull
 422 is simulated using symmetry. The meshes used in the HTC are unstructured
 423 consisting of mostly hexahedral cells and the simulations are carried out with
 424 free sink and trim. The obtained resistance coefficients are compared against
 425 three numerical references [28–30]. The simulations are carried out for a
 426 smooth hull surface i.e. surface roughness is not taken into account. This is
 427 consistent with the experiments and the other reference simulations.

Table 4: Main particulars and the grid used in the HTC. L_{pp} stands for a length between perpendiculars and N_{bl} for the number of body-fitted boundary layer cells in the wall normal direction.

Dimensions		Grid		
			Model scale (1:24)	Full scale
Length (L_{pp})	153.7 m	N_{total}	4.4×10^6	10.2×10^6
Beam (B)	27.5 m			
Draught forward (T_{fp})	9.2 m	N/T_{ap}	22	22
Draught aft (T_{ap})	10.2 m	N_{bl}	7	23
Wetted surface (S)	5454 m ²	δ_y^+	39	195

428 3.4.2. HTC results

429 The resistance coefficients from the HTC are presented and compared to
 430 model scale and full scale results in Tables 5 and 6, respectively. In the model
 431 scale, the total resistance with solvers **interFoam**, **GFMFoam** and **GFMFoam-V1**
 432 corresponds well with the experimental values. With **GFMFoam-V2**, an offset
 433 in the total resistance is noted which is attributed to the underpredicted
 434 pressure resistance. In the full scale results of Table 6, both **interFoam** and
 435 **GFMFoam** predict on average $\approx 10\%$ higher total resistance than the reference

436 simulations. We note that in case of the HTC, numerical ventilation with
437 **interFoam** is not as severe as in the Wigley hull case due to the smaller
438 near-wall aspect ratio resulting from the unstructured grid topology.

Table 5: Model scale resistance coefficients in the HTC.

	$C_p \times 10^3$	$C_f \times 10^3$	$C_t \times 10^3$	sinkage [m]	trim angle [deg]
model test			4.1	0.379	0.158
ITTC 57		2.917			
Ref. [28, 29]	0.95	2.98	3.93		
Ref. [30]	0.839	2.852	3.691		
interFoam	0.955	3.131	4.086	0.439	0.167
GFMFoam	0.958	3.096	4.054	0.418	0.168
GFMFoam-V1	0.944	3.099	4.043	0.418	0.169
GFMFoam-V2	0.553	3.069	3.622	0.410	0.190

Table 6: Full scale resistance coefficients in the HTC.

	$C_p \times 10^3$	$C_f \times 10^3$	$C_t \times 10^3$	sinkage [m]	trim angle [deg]
ITTC 57		1.497			
Ref. [28]	0.711	1.471	2.18		
Ref. [30]	0.689	1.588	2.276		
interFoam	0.779	1.685	2.464	0.437	0.161
GFMFoam	0.823	1.673	2.496	0.423	0.159

Figure 14: Stern wave heights $\frac{\zeta}{L_{pp}}$ and dynamic pressure p_d distribution on the aft body surface, model scale. I more wet transom and II higher over pressure on the hull in case GFMFoam-V2 comparing to other cases.

439 The reason for the underpredicted pressure resistance with GFMFoam-V2
 440 is illustrated in Figure 14. With GFMFoam-V2 the stern wave separates
 441 differently than with the other solvers leaving the transom clearly wet. The
 442 GFMFoam-V2 stern wave shape causes a high dynamic pressure on the aft body
 443 which in turn results in a lower pressure difference between the bow and
 444 stern with GFMFoam-V2. We note that no significant differences in the bow
 445 wave shape were observed between the studied solvers. The high pressure
 446 on the aft body also explains the clearly higher trim angle (more bow down
 447 behaviour) observed with GFMFoam-V2.

448 The model scale turbulent viscosity ratio at the free surface is shown in
 449 Figure 15. Consistent behaviour with the Wigley hull results can be seen.
 450 Higher values of ν_t are observed along the side of the hull with interFoam
 451 than with the GFM solvers. Additionally, the results of GFMFoam-V2 differ
 452 from the other GFM versions with high ν_t in the ship wake. The reason for

453 the high ν_t is likely the higher velocity gradient observed at the free surface
 454 with **GFMFoam-V2** in the shear layer test case (Figure 7). The high ν_t and the
 455 wrongly predicted velocity gradient at the wake part with **GFMFoam-V2** is the
 456 probable cause for the different wave separation visible in Figure 14.

Figure 15: Turbulent viscosity ratio at the free surface, model scale.

457 Figure 16 shows cutplanes of the velocity field with **interFoam** and **GFMFoam**
 458 in model and full scale. We again note the discontinuity in the velocity
 459 field with **interFoam** which is more pronounced in the full scale results (I)
 460 and close to the hull (II). Further, the incorrect far field wave propagation
 461 with the GFM is shown in Figure 17 for all the studied GFM solvers. We
 462 note that, although not explicitly stated by the authors, the wave pattern
 463 shown by Bakica et al. [31] using GFM indicates remarkable similarity with
 464 the present GFM results.

Figure 16: Velocity distribution on the cutplanes towards bow at positions $0.87 \times L_{pp}$, $0.94 \times L_{pp}$ and $1 \times L_{pp}$ from top to bottom. Model scale results are on the top picture and full scale (FS) in the bottom picture. 30

Figure 17: Kelvin wave with different viscous treatments, model scale. (I) Difference of a wave tip bent, (II) more damped stern wave in case `GFMFoam-V2` and (III) disturbances on the wave shape in all GFM cases comparing to `interFoam`.

4. Conclusions

In the present paper, the GFM method by Vukčević et al. was implemented and the new solver, `GFMFoam` was compared against the standard `interFoam` VOF solver in four test cases.

Based on the present steady state test cases with minor surface deformations it was noted that neither `GFMFoam` or `interFoam` clearly outperformed the other. In the Wigley hull case, worse results were obtained with `GFMFoam` on a coarse grid. However, on finer grids, `GFMFoam` results were slightly better than the `interFoam` results. At high Fr the friction resistance obtained with `interFoam` started to increase due to the unphysical turbulence viscosity generated at the free surface. The increased turbulence viscosity for `interFoam` at the free surface was noted to be likely a result of the spurious velocity generated at the free surface. Also, the `interFoam` results showed

numerical ventilation in the vicinity of the hull at high Fr . Small amplitude waves were noted to propagate better with `interFoam` when the transverse grid resolution is coarse. We note that by adjusting the interface cell identification strategy in `GFMFoam`, better wave propagation at small amplitudes could possibly be achieved. A possible avenue for improving such wave propagation could be combining the GFM with a geometric VOF, such as the `isoAdvector` approach by Roenby et al. [8], and using the sub-cell interface location consistently.

In the HTC, the viscous term treatment in the GFM affects the shape of the stern wave and thereby the pressure resistance. The pressure resistance with the `GFMFoam-V2` was noticed to be significantly underpredicted due to the incorrect wave separation at the stern. Consistent with the observations in the Wigley hull test case, we showed that `interFoam` predicts a higher turbulence viscosity and a discontinuous velocity field at the air-water interface in the HTC. Also the incorrect propagation of the GFM small amplitude waves in the far-field was observed to be consistent with the Wigley hull and HTC results.

Conflict of interest

None to declare.

Acknowledgement

The computational resources for this study were provided by CSC Finnish IT Center for Science.

References

- [1] F. Pereira, G. Vaz, L. Ea, Verification and validation exercises for the flow around the kvlcc2 tanker at model and full-scale reynolds numbers, *Ocean Engineering* 129 (2017) 133148. doi:10.1016/j.oceaneng.2016.11.005.
- [2] C. Fureby, S. Toxopeus, M. Johansson, M. Tormalm, K. Petterson, A computational study of the flow around the KVLCC2 model hull at straight ahead conditions and at drift, *Ocean Engineering* 118 (2016) 1–16. doi:10.1016/j.oceaneng.2016.03.029.
URL <https://doi.org/10.1016/j.oceaneng.2016.03.029>

- [3] A. Farkas, N. Degiuli, I. Martić, Assessment of hydrodynamic characteristics of a full-scale ship at different draughts, *Ocean Engineering* 156 (2018) 135–152. doi:10.1016/j.oceaneng.2018.03.002.
URL <https://doi.org/10.1016/j.oceaneng.2018.03.002>
- [4] H. Jasak, V. Vukčević, I. Gatin, I. Lalović, CFD validation and grid sensitivity studies of full scale ship self propulsion, *International Journal of Naval Architecture and Ocean Engineering* 11 (1) (2019) 33–43. doi:10.1016/j.ijnaoe.2017.12.004.
URL <https://doi.org/10.1016/j.ijnaoe.2017.12.004>
- [5] A. Prosperetti, G. Tryggvason, *Computational Methods for Multiphase Flow*, Cambridge University Press, 2007. doi:10.1017/cbo9780511607486.
URL <https://doi.org/10.1017/cbo9780511607486>
- [6] C. Hirt, B. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries, *Journal of Computational Physics* 39 (1) (1981) 201–225. doi:10.1016/0021-9991(81)90145-5.
URL [https://doi.org/10.1016/0021-9991\(81\)90145-5](https://doi.org/10.1016/0021-9991(81)90145-5)
- [7] D. L. Youngs, Time-dependent multi-material flow with large fluid distortion, *Numerical methods for fluid dynamics*.
- [8] J. Roenby, H. Bredmose, H. Jasak, A computational method for sharp interface advection, *Royal Society Open Science* 3 (11) (2016) 160405. doi:10.1098/rsos.160405.
URL <https://royalsocietypublishing.org/doi/abs/10.1098/rsos.160405>
- [9] F. Xiao, Y. Honma, T. Kono, A simple algebraic interface capturing scheme using hyperbolic tangent function, *International Journal for Numerical Methods in Fluids* 48 (9) (2005) 1023–1040. doi:10.1002/fld.975.
URL <https://doi.org/10.1002/fld.975>
- [10] S. M. Damián, N. M. Nigro, An extended mixture model for the simultaneous treatment of small-scale and large-scale interfaces, *International Journal for Numerical Methods in Fluids* 75 (8) (2014) 547–574.

- doi:10.1002/fld.3906.
 URL <https://doi.org/10.1002/fld.3906>
- [11] V. Vukčević, H. Jasak, I. Gatin, Implementation of the ghost fluid method for free surface flows in polyhedral finite volume framework, *Computers & Fluids* 153 (2017) 1–19. doi:10.1016/j.compfluid.2017.05.003.
 URL <https://doi.org/10.1016/j.compfluid.2017.05.003>
- [12] R. P. Fedkiw, T. Aslam, B. Merriman, S. Osher, A non-oscillatory eulerian approach to interfaces in multimaterial flows (the ghost fluid method), *Journal of Computational Physics* 152 (2) (1999) 457 – 492. doi:<https://doi.org/10.1006/jcph.1999.6236>.
 URL <http://www.sciencedirect.com/science/article/pii/S0021999199962368>
- [13] M. Kang, R. P. Fedkiw, X.-D. Liu, A boundary condition capturing method for multiphase incompressible flow, *Journal of Scientific Computing* 15 (3) (2000) 323–360. doi:10.1023/A:1011178417620.
 URL <https://doi.org/10.1023/A:1011178417620>
- [14] S. Shin, Internal wave computations using the ghost fluid method on unstructured grids, *International Journal for Numerical Methods in Fluids* 47 (3) (2005) 233–251. doi:10.1002/fld.806.
 URL <https://onlinelibrary.wiley.com/doi/abs/10.1002/fld.806>
- [15] J. Huang, P. M. Carrica, F. Stern, Coupled ghost fluid/two-phase level set method for curvilinear body-fitted grids, *International Journal for Numerical Methods in Fluids* 55 (9) (2007) 867–897. doi:10.1002/fld.1499.
 URL <https://doi.org/10.1002/fld.1499>
- [16] P. Queutey, M. Visonneau, An interface capturing method for free-surface hydrodynamic flows, *Computers & Fluids* 36 (9) (2007) 1481–1510. doi:10.1016/j.compfluid.2006.11.007.
 URL <https://doi.org/10.1016/j.compfluid.2006.11.007>
- [17] S. S. Deshpande, L. Anumolu, M. F. Trujillo, Evaluating the performance of the two-phase flow solver interFoam, *Computational Science &*

- 575 Discovery 5 (1) (2012) 014016. doi:10.1088/1749-4699/5/1/014016.
576 URL <https://doi.org/10.1088/1749-4699/5/1/014016>
- 577 [18] H. G. Weller, G. Tabor, H. Jasak, C. Fureby, A tensorial approach to
578 computational continuum mechanics using object-oriented techniques,
579 Comput. Phys. 12 (6) (1998) 620. doi:10.1063/1.168744.
580 URL <https://doi.org/10.1063/1.168744>
- 581 [19] C. Yang, F. Huang, L. Wang, D. cheng Wan, Numerical simulations
582 of highly nonlinear steady and unsteady free surface flows, Journal of
583 Hydrodynamics 23 (6) (2011) 683–696. doi:10.1016/s1001-6058(10)
584 60166-7.
585 URL [https://doi.org/10.1016/s1001-6058\(10\)60166-7](https://doi.org/10.1016/s1001-6058(10)60166-7)
- 586 [20] R. si Zha, H. xuan Ye, Z. rong Shen, D. cheng Wan, Numerical compu-
587 tations of resistance of high speed catamaran in calm water, Journal of
588 Hydrodynamics 26 (6) (2014) 930–938. doi:10.1016/s1001-6058(14)
589 60102-5.
590 URL [https://doi.org/10.1016/s1001-6058\(14\)60102-5](https://doi.org/10.1016/s1001-6058(14)60102-5)
- 591 [21] D. W. Park, S. B. Lee, The sensitivity of ship resistance to wall-
592 adjacent grids and near-wall treatments, International Journal of Naval
593 Architecture and Ocean Engineering 10 (6) (2018) 683–691. doi:
594 10.1016/j.ijnaoe.2017.12.003.
595 URL <https://doi.org/10.1016/j.ijnaoe.2017.12.003>
- 596 [22] S. Muzaferija, M. Peri'c, COMPUTATION OF FREE-SURFACE
597 FLOWS USING THE FINITE-VOLUME METHOD AND MOVING
598 GRIDS, Numerical Heat Transfer, Part B: Fundamentals 32 (4) (1997)
599 369–384. doi:10.1080/10407799708915014.
600 URL <https://doi.org/10.1080/10407799708915014>
- 601 [23] R. C. LOCK, THE VELOCITY DISTRIBUTION IN THE LAMINAR
602 BOUNDARY LAYER BETWEEN PARALLEL STREAMS, The Quar-
603 terly Journal of Mechanics and Applied Mathematics 4 (1) (1951) 42–63.
604 doi:10.1093/qjmam/4.1.42.
605 URL <https://doi.org/10.1093/qjmam/4.1.42>
- 606 [24] H. Kajitani, H. Miyata, M. Ikehata, H. Tanaka, H. Adachi, M. Nami-
607 matsu, S. Ogiwara, The summary of the cooperative experiment on

- 608 wigley parabolic model in japan, Tech. rep., TOKYO UNIV (JAPAN)
609 (1983).
- 610 [25] P. Peltonen, Meshing script for Wigley hull, https://github.com/hamsteri15/OpenFOAM/tree/master/meshing_scripts/wigleyHull,
611 [Online; accessed 27-April-2020] (2019).
- 613 [26] B. E. Larsen, D. R. Fuhrman, On the over-production of turbulence be-
614 neath surface waves in reynolds-averaged navier–stokes models, *Journal*
615 *of Fluid Mechanics* 853 (2018) 419–460. doi:10.1017/jfm.2018.577.
616 URL <https://doi.org/10.1017/jfm.2018.577>
- 617 [27] I. R. Procedures, Guidelines 7.5-03-02-02, Resistance Test, Revision 3.
618 URL <https://ittc.info/media/1932/75-03-02-02.pdf>
- 619 [28] S. Hänninen, Evaluation of the scale effect on the flow around a ship
620 hull using cfd, masters thesis.
- 621 [29] S. Hänninen, J. Sehweighofer, Numerical investigation of the scale effect
622 on the flow around a ship hull, *Ship Technology Research* 53 (1) (2006)
623 17–25. doi:10.1179/str.2006.53.1.004.
624 URL <https://doi.org/10.1179/str.2006.53.1.004>
- 625 [30] H. C Raven, A. Ploeg, A. R Starke, Towards a cfd-based prediction of
626 ship performance - progress in predicting full-scale resistance and scale
627 effects, RINA - International Conference - Marine CFD 2008 - Papers.
- 628 [31] A. Bakica, I. Gatin, V. Vukčević, H. Jasak, N. Vladimir, Accurate assess-
629 ment of ship-propulsion characteristics using CFD, *Ocean Engineering*
630 175 (2019) 149–162. doi:10.1016/j.oceaneng.2018.12.043.
631 URL <https://doi.org/10.1016/j.oceaneng.2018.12.043>
- 632 [32] L. Lobovský, E. Botia-Vera, F. Castellana, J. Mas-Soler, A. Souto-
633 Iglesias, Experimental investigation of dynamic pressure loads dur-
634 ing dam break, *Journal of Fluids and Structures* 48 (2014) 407–434.
635 doi:10.1016/j.jfluidstructs.2014.03.009.
636 URL <https://doi.org/10.1016/j.jfluidstructs.2014.03.009>

637 **Appendix A. Computation of the explicit pressure gradient in**
638 **GFMFoam**

A linear interpolation (without interface correction) of pressure from cell center values p_P and p_N to faces p_f reads

$$p_f = f_x p_P + (1 - f_x) p_N, \quad (\text{A.1})$$

639 where f_x are the grid weights. The interpolation formula in Eq. A.1 can be
640 directly used to form the gradient without interface correction as shown in
641 Algorithm 1.

Algorithm 1 Computation of the standard pressure gradient

```

1:  $p_f \leftarrow \text{Eq. A.1}$  ▷ Standard interpolation from cells to faces
2: for  $f \leftarrow \text{all face indices}$  do
3:    $\text{cell index } P \leftarrow \text{owner cells}[f]$ 
4:    $\text{cell index } N \leftarrow \text{neighbour cells}[f]$ 
5:    $\nabla p[P] += p_f[f] \times \vec{n}[f]$ 
6:    $\nabla p[N] -= p_f[f] \times \vec{n}[f]$ 
7: end for
8:  $\nabla p \leftarrow \nabla p / V$  ▷ Divide by the cell volumes

```

In the GFMFoam pressure gradient computation an estimate for the interface location between the cell centers is first required

$$\vec{x}_\Gamma = \vec{x}_P + \lambda \vec{d}, \quad (\text{A.2})$$

where $\vec{x}_P$ is the location of cell center P and $\vec{d}$ is a vector from P to N and λ is a weighting parameter

$$\lambda = \frac{\alpha_P - 0.5}{\alpha_P - \alpha_N}. \quad (\text{A.3})$$

Using $\vec{x}_\Gamma$ a hydrostatic jump in the pressure values across the interface can be computed as

$$H = (\rho_L - \rho_G) g \vec{x}_\Gamma. \quad (\text{A.4})$$

642 The algorithm for the interface corrected pressure gradient is shown in
643 Algorithm 2.

Algorithm 2 Computation of GFM pressure gradient.

```

1:  $p_f \leftarrow$  Eq. A.1 ▷ Standard interpolation from cells to faces
2: for  $f \leftarrow$  all face indices do
3:   cell index  $P \leftarrow$  owner cells[ $f$ ]
4:   cell index  $N \leftarrow$  neighbour cells[ $f$ ]
5:   if  $(\alpha_P - 0.5)(\alpha_N - 0.5) < 0$  then ▷ Interface cell pair
6:      $\lambda \leftarrow$  Eq. A.3
7:      $\vec{x}_\Gamma \leftarrow$  Eq. A.2 ▷ Interface location
8:      $H \leftarrow$  Eq. A.4 ▷ Pressure jump condition
9:     if  $\alpha[P] > 0.5$  then ▷ cell  $P$  is wet and  $N$  dry
10:       $p^+ \leftarrow$  Eq. A.5
11:       $p^- \leftarrow$  Eq. A.5
12:    end if
13:    if  $\alpha[P] \leq 0.5$  then ▷ cell  $P$  is dry and  $N$  wet
14:       $p^+ \leftarrow$  Eq. A.6
15:       $p^- \leftarrow$  Eq. A.6
16:    end if
17:     $\beta \nabla p[P] += \vec{n}[f] \times (f_x[f]p[P] + (1 - f_x[f])p^-)$ 
18:     $\beta \nabla p[N] -= \vec{n}[f] \times (f_x[f]p^+ + (1 - f_x[f])p[N])$ 
19:  else ▷ Not an interface cell pair
20:     $\beta \nabla p[P] += \beta[P] \times p_f[f] \times \vec{n}[f]$ 
21:     $\beta \nabla p[N] -= \beta[N] \times p_f[f] \times \vec{n}[f]$ 
22:  end if
23: end for
24:  $\nabla p \leftarrow \nabla p / V$  ▷ Divide by the cell volumes

```

Following the notation in Vukčević et al. [11] the ghost cell values $p_P^\pm$ and $p_N^\pm$ for the case when cell P is wet (N is dry) are

$$\begin{aligned}
p_N^+ &= \frac{\beta^-}{\beta_w} p_N + \left(1 - \frac{\beta^-}{\beta_w}\right) p_P - \frac{\beta^-}{\beta_w} H \\
p_P^- &= \frac{\beta^+}{\beta_w} p_P + \left(1 - \frac{\beta^+}{\beta_w}\right) p_N + \frac{\beta^+}{\beta_w} H.
\end{aligned} \tag{A.5}$$

For a dry cell P (and wet N)

$$\begin{aligned} p_N^- &= \frac{\beta^+}{\widehat{\beta}_d} p_N + \left(1 - \frac{\beta^+}{\widehat{\beta}_d}\right) p_P + \frac{\beta^+}{\widehat{\beta}_d} H \\ p_P^+ &= \frac{\beta^-}{\widehat{\beta}_d} p_P + \left(1 - \frac{\beta^-}{\widehat{\beta}_d}\right) p_N - \frac{\beta^-}{\widehat{\beta}_d} H. \end{aligned} \tag{A.6}$$

644 In Eq. A.5 and A.6 $\beta^+ = 1/\rho_L$, $\beta^- = 1/\rho_G$, $\widehat{\beta}_d = \lambda\beta^+ + (1 - \lambda)\beta^-$ and
645 $\widehat{\beta}_w = \lambda\beta^- + (1 - \lambda)\beta^+$.

646 Appendix B. User input parameters of the localEuler scheme

Table B.7: Input parameters for the localEuler scheme.

parameter	value
maxCo	3
maxAlphaCo	1.5
rDeltaTSmoothingCoeff	0.05
rDeltaTDampingCoeff	0.5
nAlphaSpreadIter	2
nAlphaSweepIter	2
maxDeltaT	1

647 Appendix C. Two-dimensional breaking of a dam

648 To test the performance of the solvers in a test case with strong free
649 surface deformation, we consider the two-dimensional dam break problem.
650 The dimensions of the domain are presented in Figure C.18 where H1-H4
651 correspond to the locations of measurement probes for which experimental
652 wave height data is provided by Lobovský et al. [32]. The domain is a
653 1610 mm $\times$ 1610 mm square with an initial water height $h_0 = 300$ mm set in
654 the bottom right corner. The simulation is run on a equidistant $N_x \times N_y =$
655 100 $\times$ 100 Cartesian grid with a density ratio of 1000 and $g = 9.81$ m/s².
656 Due to the transient nature of the dam break problem, the implicit first order
657 Euler time integration method is used in the dam break.

Figure C.18: Dimensions (in millimeters) of the domain in the dam break problem. H1 - H4 indicate locations from which experimental wave height data is provided by Lobovský et al. [32].

658 The results for the wave height at locations H1-H4 are presented in Fig-
659 ure C.19. For locations H2-H4 two waves can be distinguished, the primary
660 wave and the secondary wave. The primary wave is the one reaching the
661 locations H2-H4 first (for instance at $t(g/h_0)^{1/2} \approx 1$ for H2) when the bulk
662 of the water moves to $x < 0$ direction. The secondary wave issues from the
663 impact to the left wall of the domain when the bulk of the water moves to
664 $x > 0$ direction (for instance at $t(g/h_0)^{1/2} \approx 7$ for H2). For H1, only the
665 secondary wave can be distinguished. For the arrival of the primary wave,

666 both solvers produce qualitatively similar results. Considering the secondary
 667 wave, **interFoam** shows an earlier arrival at all locations. The difference be-
 668 tween the two solvers is largest at the location furthest away from the left
 669 wall (H1) where the secondary wave arrives last. We note that the shape of
 670 the secondary wave is poorly captured with both solvers. This is very likely
 671 caused by the neglect of three-dimensional turbulence effects in the present
 672 dam break setup.

Figure C.19: The water height at locations H1-H4 as a function of time.

673 When comparing the shape of the wave front in the early stages of the dam
 674 break simulation (see Figure C.20), we note that **interFoam** and **GFMFoam**

675 produce a different shape. The `interFoam` wave front shape is sharp while
 676 `GFMFoam` shows a more blunt wave front shape. However, both wave fronts
 677 seem to be moving with close to same speed as the primary wave arrives to
 678 locations H2-H4 almost at the same time in Figure C.19.

679 The effect of setting an $\alpha = 0.49$ initial condition for `GFMFoam` is also
 680 illustrated in Figure C.20. Since no cell pairs for which $(\alpha_P - 0.5)(\alpha_N -$
 681 $0.5) < 0$ are found in the domain the initial condition remains throughout
 682 the simulation in `GFMFoam`.

Figure C.20: Illustrations of the different wave front shapes with `interFoam` and `GFMFoam` (top) and the effect of the $\alpha = 0.49$ initial condition (bottom).