Ocean Engineering 121 (2016) 76–88

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Decomposition model for naval hydrodynamic applications, Part II: Verification and validation

Vuko Vukčević [a,*], Hrvoje Jasak [a,b], Šime Malenica [c]

[a] University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, Zagreb, Croatia
[b] Wikki Ltd, 459 Southbank House, SE1 7SJ London, United Kingdom
[c] Bureau Veritas, 67/71, 92200 boulevard du Château, Neuilly-sur-Seine, France

ARTICLE INFO

Article history:
Received 21 April 2015
Received in revised form
22 March 2016
Accepted 10 May 2016

Keywords:
Decomposition model
Numerical wave tank
Vertical cylinder
Higher-order wave loads
OpenFOAM

ABSTRACT

This paper presents the verification and validation of the decomposition computational model presented in the accompanying paper. The decomposition model is primarily targeted to simulating incompressible, free surface flows in naval hydrodynamics related to wave-like phenomena. The model is implemented in foam-extend-3.1, a community driven fork of the OpenFOAM CFD software. The first set of tests considers regular wave propagation in a 2-D tank. Studies regarding Level Set parameters and reflection are carried out. Mass conservation during a long simulation is also considered. The solution is compared with non-linear potential flow solution obtained with stream-function wave theory for a range of wave steepness parameters. Higher-order wave loads on a vertical circular cylinder are calculated and compared to well-established non-linear potential flow results. Time refinement, mesh refinement and wave frequency studies are carried out and presented.

1. Introduction

This paper presents the verification and validation of the computational model presented in the accompanying paper. The model is based on the Spectral Wave Explicit Navier–Stokes (SWENSE) method (Ferrant et al., 2002; Ducrozet et al., 2014; Monroy et al., 2010; Marcer et al., 2007) with implicit relaxation zones (Jasak et al., 2015) in the far field to prevent wave reflection. The interface is captured with an implicitly redistanced Level Set (LS) method derived from the phase field (PF) equation (Sun and Beckermann, 2007, 2008). The solution algorithm is based on a combination of SIMPLE (Patankar and Spalding, 1972) and PISO (Issa, 1986) algorithms and is implemented in foam-extend-3.1, a community driven fork of the OpenFOAM (Weller et al., 1998), which is an Open Source C++ library for computational continuum mechanics.

A progressive increase in computer resources made Reynolds-Averaged Navier–Stokes (RANS) models available for both scientific and industrial use. RANS methods closely model the governing physics and often give detailed information on the flow field. Nevertheless, all computational methods need to be tested against well-established results prior to their general use.

Verification and validation of CFD-based methods in naval hydrodynamics is still under-way. Larsson et al. (2013) provided an excellent assessment of the various state-of-the-art Computational Fluid Dynamics (CFD) codes for both steady-state and transient calculations. Queutey and Visonneau (2007) used Volume of Fluid (VOF) based Finite Volume (FV) solver on a Series 60 ship hull. They have shown excellent agreement with experimental results illustrated by comparisons of free-surface elevations and velocity field. However, they have reported noticeable differences in the solution obtained with a grid of 3 million and 4 million computational points. Jasak et al. (2014) developed a rapid steady-state resistance solver based on the FV method in OpenFOAM. The solver has been validated with Kriso Container Ship (KCS) and US Navy Combatant DTMB 5415 where drag force coefficients and free-surface elevation were compared to experimental data. Although the total drag force was within 2% compared to experiments, mesh convergence for the KCS case proved to be oscillatory within narrow band. Presented examples illustrate the need for mesh refinement studies in CFD.

Transient calculations of wave loads are traditionally done in the frequency domain (Malenica and Molin, 1995) assuming potential flow. Potential flow methods provide an extremely fast insight in the problem at hand, but have difficulty tackling phenomena where the rotational and viscous effects are significant. For above-mentioned reasons, wave modelling using the RANS approach is gaining popularity.

Higuera et al. (2013a, 2013b) presented a RANS method based on dynamic boundary conditions used for wave generation and absorption. Domain is not decomposed in this approach, and the

* Corresponding author.
E-mail addresses: vuko.vukcevic@fsb.hr (V. Vukčević), h.jasak@wikki.co.uk, hrvoje.jasak@fsb.hr (H. Jasak), sime.malenica@bureauveritas.com (Š. Malenica).

http://dx.doi.org/10.1016/j.oceaneng.2016.05.021
0029-8018/

RANS solution is calculated in the whole domain. Similarly, Luppes et al. (2012) use absorbing boundary conditions for wave modelling on Cartesian, block structured grids with cell cutting algorithms to model both the solid bodies and the free surface. Other authors focussed on decomposition models. Jacobsen et al. (2012) introduced a domain decomposition through explicit relaxation zones for one-way coupling between RANS solution and potential flow wave theories. Using the same domain decomposition approach, Paulsen et al. (2014a, 2014b) presented a coupling with non-linear potential flow solution that has been used to obtain wave loads on vertical circular cylinders. Ferrant et al. (2002) initially introduced the solution decomposition strategy for extremely efficient coupling with advanced potential flow wave theories where the wave reflection is prevented with extremely coarse computational grid in the far-field. A large portion of this work shows a good agreement of RANS simulations with experiments. Nevertheless, mesh refinement studies are scarce, while the time resolution studies are usually not performed.

In this paper, verification and validation of the method described in part 1 starts with two-dimensional (2-D) regular wave propagation. Influence of the diffusion parameter in the LS transport equation, b is assessed. Relaxation zone length is varied in order to determine minimum length without significant wave reflection. Results for a series of waves with different steepness are compared to non-linear stream function wave theory (Rienecker and Fenton, 1981). A long run has been executed to show that the mean water level rise does not occur as indicated by Higuera et al. (2013a) for algorithms which use dissipative zones. This also tested the conservation properties of the SWENSE decomposed, implicitly redistanced LS method. A long domain (approximately 8 wave lengths) is used to assess numerical diffusion effects on wave propagation. Three-dimensional (3-D) test cases simulating regular wave loads on a surface-piercing vertical cylinder are also presented. A time refinement study for a representative case has been performed to assess the numerical error related to time resolution. A mesh refinement study has been performed to assess the error related to spatial resolution. Then, a series of simulations with regular waves of varying wave numbers are carried out. Higher-order in-line forces are compared to fully non-linear potential flow solution by Ferrant et al. (1999). For all test cases, non-uniform meshes with high aspect ratio cells are used to achieve cell clustering towards the free surface and object of interest.

2. Numerical wave tank

The first test case deals with 2-D wave propagation in a numerical wave tank. Simulation parameters and geometry characteristics are presented for a benchmark case. The benchmark case results are compared with the non-linear potential flow stream function wave theory (Rienecker and Fenton, 1981). Influence of the diffusion parameter in the LS transport equation, b is assessed through LS Courant–Friedrichs–Lewy number, CFL_ψ and stabilisation constant, γ. Next, a reflection study will be carried out by changing the relaxation zone length, and is followed by a wave steepness study. The variation of wave steepness is achieved by varying the wave height while keeping the wave period constant. A long simulation has been carried out in order to assess the conservative properties of implicit relaxation zones and the LS method. A simulation with longer domain is done to examine the effects of numerical diffusion on the wave propagation.

2.1. Benchmark case

For the benchmark case, a wave with mild steepness, $ka \approx 0.023$ is simulated. Wave and simulation parameters are given in Table 1.

Table 1
Wave and simulation parameters of the benchmark case.

Wave height	H, m	0.1
Wave period	T, s	3
Wave frequency	ω, rad/s	2.0944
Wave length	λ, m	13.934
Wave number	k, m	0.450924
Depth	d, m	6
Relaxation zone length	λ_r, m	22.5
Courant–Friedrichs–Lewy number	CFL	0.125
LS Courant–Friedrichs–Lewy number	CFL_ψ	0.25
Stabilisation constant	γ	100 000
Smearing distance	ϵ	0.004

The bounding box of the block-structured, hexahedral mesh is $x \in [0, 60]$ and $y \in [-6, 0.3]$ m. The mesh consists of three longitudinal and three vertical blocks. The cells are heavily graded towards the middle block which is 15 m $(\approx \lambda)$ long (Fig. 1) and 0.2 m high $(\approx 2H)$ (Fig. 2). In the middle block, there are approximately 15 cells per wave height and 100 cells per wave length. This results in maximum cell aspect ratio of 22.2:1. The resulting mesh may be considered relatively coarse considering the number of cells per wave height and relatively fine considering the number of cells per wave length in the middle block. Overall, 11 700 cells is considered coarse from a computational perspective, making the mesh suitable for various sensitivity studies.

Relaxation zones are positioned at inlet and outlet (far field) boundaries. The length of relaxation zones is 22.5 m, or $\lambda_r \approx 1.5\lambda$. Hence, a full CFD solution is obtained in the middle of the domain, spanning 15 m in the longitudinal direction. This is presented in the top part of Fig. 3. This region represents only 25% of the domain. Nevertheless, the exponential character of the weight field (Jacobsen et al., 2012):

$$w = \frac{e^{\left(\frac{d}{\lambda}\right)^p} - 1}{e - 1}, \tag{1}$$

is favourable, because a part of the solution that is between 50 and 100% CFD is present in 86.5% of the domain (see the middle image of Fig. 3). For example, if w=0.5, the solution is a blend of 50% CFD solution and 50% potential flow solution. If one uses advanced potential flow wave theories, this solution is also valid. In this example, 96.6% of the cells are in this region, so the cell count is only slightly increased.

Wave gauges are positioned in the middle part where the full CFD solution is achieved. The longitudinal coordinates are 25, 30 and 35 m for wave gauges 1, 2 and 3, respectively. The simulation time is 30 s which corresponds to 10 periods. Time evolution of the wave elevation at wave gauges is presented in Fig. 4. Only the last (representative) period is shown for clarity. A good agreement between CFD and stream function wave theory is presented. Peaks and troughs remain within 0.6% of relative error defined as:

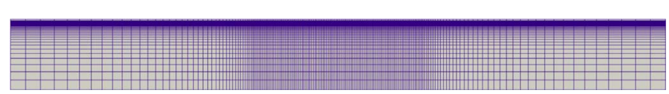

Fig. 1. Full view of the mesh.

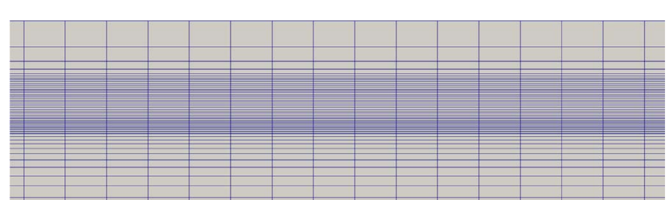

Fig. 2. Zoomed view of the mesh in the middle of the domain, near the free surface.