IX International Conference on Computational Methods in Marine Engineering
MARINE 2021

FULLY REPRODUCIBLE RANS SHIP HYDRODYNAMICS FOR THE JBC VALIDATION CASE

M. Liefvendahl*, L. Sjökvist* AND M. Winroth*

*Swedish Defense Research Agency (FOI)
164 90 Stockholm, Sweden
e-mail: mattias.liefvendahl@foi.se, web page: https://www.foi.se

Key words: Ship Hydrodynamics, RANS, Validation

Abstract. Simulation results are presented for a well established ship hydrodynamics validation case with the Japan Bulk Carrier (JBC). The results include the ship position, forces on the hull, water surface deformation and the stern flow. Simulation results are compared with measurements for all these quantities. The open source software OpenFOAM was employed, with finite volume numerics, RANS turbulence modelling, the volume-of-fluid method for the free surface, and ship motion functionality. In order to enhance the reproducibility of the results, the data files of the simulation case are made freely available. In combination with open source software, this allows for other research groups to re-simulate, modify and improve the case. Practical aspects of making this type of simulation data available are also discussed in the paper.

1 INTRODUCTION

There are two objectives with this paper. The first is to provide new simulation results for a well-known ship hydrodynamic validation case, the Japan Bulk Carrier (JBC) in towed condition. The second objective is to enhance the reproducibility of results by taking full advantage of the open-source concept, and making simulation data directly and freely available for other research groups. Data selection and procedures for this are discussed in a separate section of the present paper.

The selected case is the JBC in towed condition, which was one of the test cases of the Tokyo 2015 workshop on CFD in ship hydrodynamics, [4]. The ship model was designed for CFD validation, and also for testing with and without an energy saving device. Here the hull without energy saving device was selected. No full-scale ship exists, but the design corresponds to a cape-size bulk carrier with length between perpendiculars, $L_{pp} = 280.0\,\mathrm{m}$. We emphasize that all investigations presented here are carried out in model scale, corresponding to the available measurement data as described in section 2 where all parameters are given. The JBC model is shown in figure 1.

New simulations have been carried out using RANS turbulence modelling based on the $k-\omega$ SST model, the volume-of-fluid (VoF-)method for the free surface, and a ship model free to heave

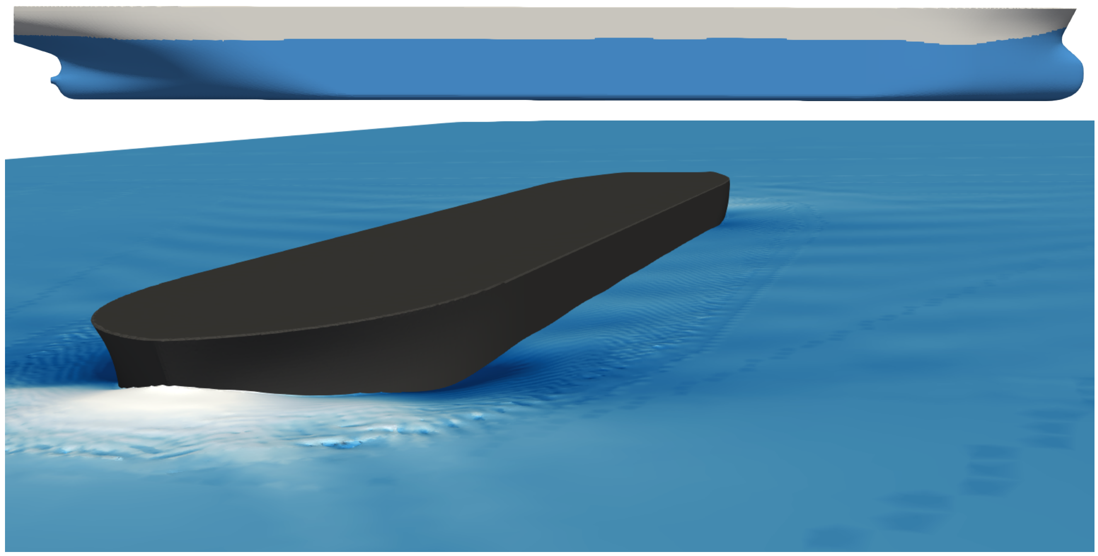

Figure 1: Simulation results for the JBC at, Fr = 0.142. Top: Side view of the hull, with the wetted part coloured in blue. Bottom: Perspective view with the water surface coloured by elevation.

and pitch. A key aspect for reproducibility of results is the employed `interDyMFoam`-solver, [2], in the open-source software package OpenFOAM v5.0[1]. The grid generation, the modelling and the numerics are described in section 3. Results will be compared with measurements, [3, 4], for resistance, trim and sinkage, wave cuts along the hull, and the flow in the propeller region. The results and the validation are presented in section 4.

Practical aspects of making simulation data available, for results reproduction by other researchers, are discussed in section 5. In particular, the choices of which data to select are discussed. With a minimal data set, it is possible to regenerate the mesh and carry out the simulation. It may however be practical to include also simulation results so that post-processing can immediately be done and also to double check that the results obtained in the re-simulation actually matches the published data. The data repository which was chosen is also indicated. Based on this information, it should be possible for other research groups to fully reproduce all results presented in this paper. Generally, for OpenFOAM, there is a good set of tutorial cases available for all different solvers. There is however a strong need to also make validation cases available, and this paper aims to contribute to this effort.

2 THE JAPAN BULK CARRIER TEST CASE

Here all the information and parameters for the selected test case are provided. The ship model is tested without rudder or energy saving device in towed condition (hence without propeller). The model is shown in figure 1 and the parameters are given in table 1. The main focus is on the design Froude number, Fr = 0.142, but additional simulations were also carried out at different speeds and results for trim, sinkage and resistance as function of speed (or Froude/Reynolds number) are provided.

[1]https://openfoam.org

Table 1: Parameters for the JBC simulation at Fr = 0.142, corresponding to the experiments carried out at NMRI (National Maritime Research Institute, Japan), see p.27 of [4]. The Reynolds number is, $\mathrm{Re} = 7.42 \cdot 10^6$.

Quantity	Notation	Value	Unit
Length between perpendiculars	L_{pp}	7 000	mm
Beam at water line	B_{wl}	1 125	mm
Draft	T	413	mm
Wet surface	S	12.223	m^2
Displacement	Δ	2.787	m^3
Velocity	V_0	1.177	m/s
Kinematic viscosity water	ν_w	$1.11 \cdot 10^{-6}$	m^2/s
Kinematic viscosisty air	ν_a	$1.54 \cdot 10^{-5}$	m^2/s
Water density	ρ_w	998.2	$\mathrm{kg/m}^3$
Air density	ρ_a	1.2	$\mathrm{kg/m}^3$
Gravitational acceleration	g	9.81	$\mathrm{m/s}^2$

Validation data is freely available from the Tokyo workshop homepage[2] and it is also described in [4]. The data include resistance, trim and sinkage, wave cuts parallel to the hull, and also the mean velocity in cross-planes in the stern region.

3 METHODS AND PRE-PROCESSING

The two phases, water and air, are simulated using the volume-of-fluid (VoF) method, [5]. A phase indicator function, $\alpha(\mathbf{x}, t)$, is used and the flow is described as one continuum with density and viscosity given by the following.

$$\rho = \alpha\rho_w + (1-\alpha)\rho_a \qquad \nu = \alpha\nu_w + (1-\alpha)\nu_a$$

Here, $\mathbf{x}$ is the spatial coordinate vector, t is time and sub-scripts w and a denote water and air respectively. A bold-face font is used to indicate vectors. The incompressible flow equations are complemented with a transport equation for α, leading to five scalar equations (before turbulence modelling) for the five unknowns; three components of velocity ($\mathbf{v}$), pressure (p), and α.

The ship is allowed to pitch and heave, and thus has two degrees of freedom of motion. The computations are time-resolved, but the aim is to find the steady state equilibrium solution. Hence, in order to shorten transients, damping factors are used for the hull movement. The grid is body-fitted and deforms during the motion to adapt to the changing position of the hull.

The turbulence is modelled by the $k-\omega$ SST model, [6]. A wall-function is used for the turbulent boundary layer over the hull. Since the simulation is time-resolved, for the water surface dynamics, the simulations are run as unsteady RANS. For the simulations presented here, the `interDyMFoam`-solver, [2], in OpenFOAM v5.0 was used. The spatial discretisation is based on the finite-volume method.

The coordinate system has the x-axis pointing from bow to stern, with $x = 0$ at the forward perpendicular (bow), the z-axis pointing upwards, with $z = 0$ at the (undisturbed) water surface,

[2]https://t2015.nmri.go.jp

and y-axis to complete a right-handed system, with $y = 0$ on the symmetry plane of the hull. The simulation case actually employs another cartesian coordinate system, but in this paper the same coordinates are used as for measurements, [4], for ease of reference. A rectangular computational domain is employed which extends to, $y = \pm 1.6 L_{pp}$, and $z = \pm 2.1 L_{pp}$. The inflow patch is located $1.6 L_{pp}$ upstream of the bow, and the distance from the stern and the outflow patch is $4.2 L_{pp}$. A computational grid with $9.56 \cdot 10^6$ polyhedral cells was generated using the OpenFOAM utility `snappyHexMesh`. An adjustable time step is used, to keep the Courant number below a threshold. The time-step is rather short, on average about 1 ms, leading to around 6 000 time steps per hull flow-pass time, L_{pp}/V_0.

4 RESULTS

In this section, simulation results are compared to measurement data and the level of agreement is discussed. The force coefficients are obtained by the following standard normalization of any force, F,

$$C = \frac{F}{\rho S V_0^2/2},$$

see table 1 for notation. The following standard notation in terms of sub-scripts is used for the force coefficients. Sub-script "t" for *total* resistance, "p" for contribution of *pressure* forces to the resistance, "v" for contribution of *viscous* forces, and "r" for the residual resistance according to the ITTC definition, [1]. The relation between the total and residual coefficients is,

$$C_t = (1+k) C_f(\mathrm{Re}) + C_r,$$

where $C_f(\mathrm{Re})$ is the ITTC-1957 friction line and k is the hull form factor. The form factor determined at NMRI is, $k = 0.314$, see table 5 in [4]. In table 2, the coefficients at design Froude number are given, including measurement results both from NMRI and Osaka University (OU). For the simulations, C_r was calculated from the computed C_t, $C_f(\mathrm{Re})$ and the NMRI-value of the form factor.

The ship equilibrium position is described by its pitch (or trim) and the sinkage. The pitch angle, θ, is defined positive with bow down. The sinkage, Δz, is the vertical displacement of the centre of mass, defined positive upwards. The results for pitch and sinkage are also given in table 2.

Table 2: Force coefficients, pitch and sinkage at Fr = 0.142. The measurement results are taken from the table on page 30 of [4]. The Reynolds number is $7.42 \cdot 10^6$, except for the shorter OU-model for which, $\mathrm{Re} = 2.17 \cdot 10^6$.

	$10^3 C_v$	$10^3 C_p$	$10^3 C_r$	$10^3 C_t$	$\theta(^o)$	$10^3 \Delta z / L_{pp}$
Simulation	3.39	1.38	0.61	4.76	0.11^o	-1.00
Meas. NMRI	- - -	- - -	0.15	4.29	0.10^o	-0.86
Meas. OU	- - -	- - -	0.25	5.27	0.10^o	-0.77

In addition to the design condition, Fr = 0.142, simulations were carried out at four additional velocities, and the resulting resistance coefficients are plotted versus Froude number in figure 2. Included in that plot is also the measured total resistance, C_t, and the ITTC-1957 friction curve.

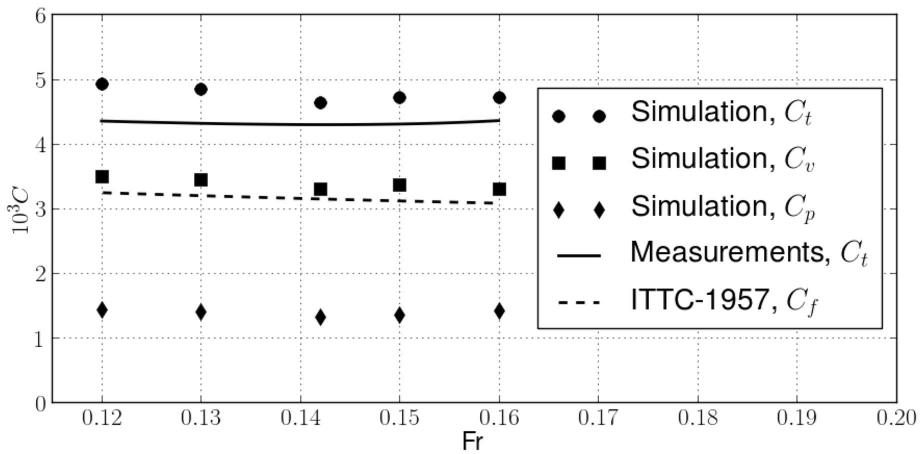

Figure 2: Resistance coefficients as functions of Froude number. The measured total resistance, C_t, is from NMRI, [3].

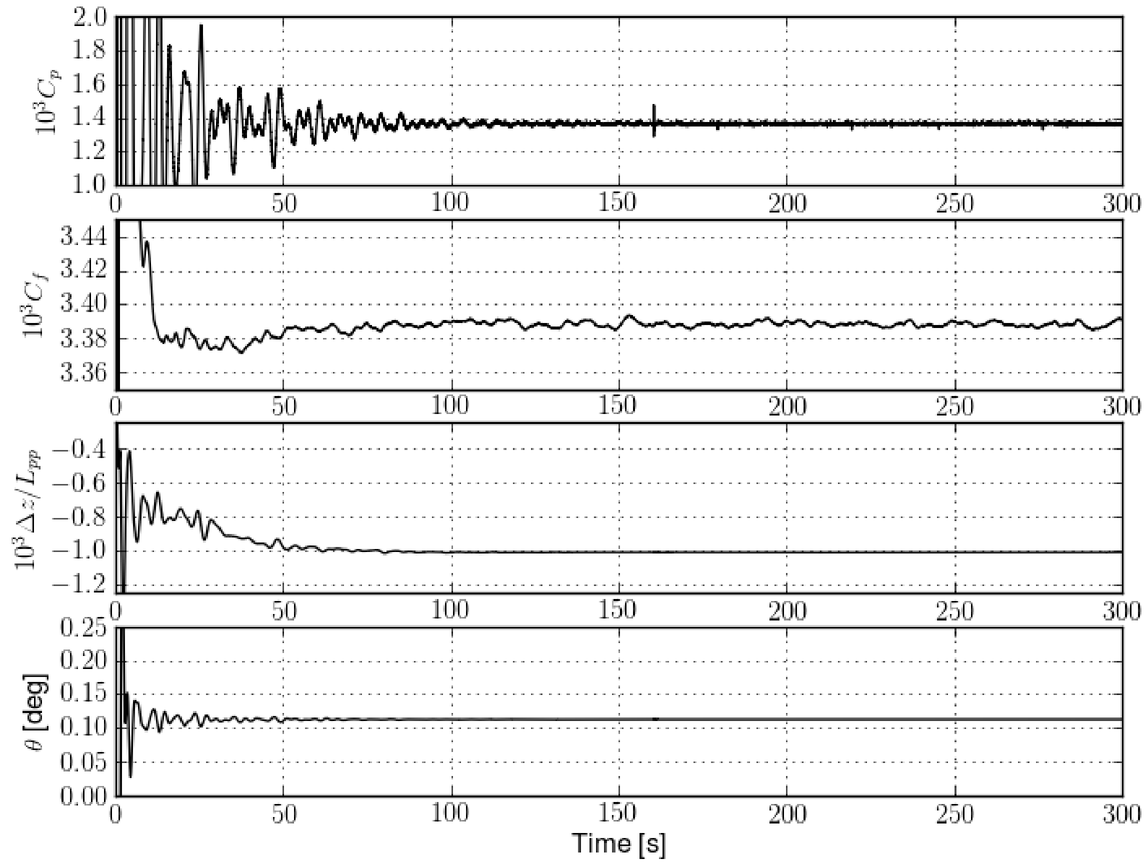

Figure 3: Time history of the pressure and viscous drag coefficients, as well as sinkage and pitch angle.

The simulations are time-resolved and in figure 3, the time history of the integral quantities is plotted. It is seen that steady-state has been reached at, $t \approx 150\,\mathrm{s}$, to be compared with the ship flow-pass time, $L_{pp}/V_0 = 5.95\,\mathrm{s}$. The simulation was restarted at, $t = 160\,\mathrm{s}$, and as a result of this, a small spike in the C_p-curve is seen at this time instant. The initial transients are mainly related to two effects; (i) The motion of the ship as it converges to its equilibrium position, and; (ii) The formation of the steady wave pattern around the ship, as the initial transient waves are propagated away. As is clear from figure 3, the pitch reaches a steady value at, $t \approx 60\,\mathrm{s}$, and the sinkage at $t \approx 90\,\mathrm{s}$. For the force coefficients it takes slightly longer to approach steady state (or quite small oscillations around the equilibrium state).

Next, the water surface predictions are discussed. In figure 4, a comparison of water surface elevation with measurements is plotted. Note that the measurements were carried out for a model towed at NMRI in *trim-free* condition at $\mathrm{Fr} = 0.142$, [4]. Because of the very small pitch angle of the simulations, $\theta = 0.11^o$, the effect of this difference on the wave pattern is believed to be small. Generally, the agreement between simulations and measurements for the wave pattern is relatively good. For example, the shorter waves along the hull, with wave length $\approx 0.1L_{pp}$, are captured in the simulations. The amplitude (peak to trough) of these waves is approximately $L_{pp}/1000$ along the wave cut at, $y = 0.19L_{pp}$. Visible discrepancies include the slight difference in the wave trough depth along the hull at the bow and the slight axial shift of the forward part of the wave cut at $y = 0.1043L_{pp}$. In addition, some spurious numerical (mesh-related) effects are seen in the water surface prediction. In particular, small artifical ripples at a distance of $\approx L_{pp}/10$ from the hull, where there is a change in mesh resolution.

The final results comparison concerns the flow in the stern region. Simulation results are compared with stereoscopic particle image velocimetry (SPIV) measurements (by NMRI, [4]) in figures 5 and 6. The measurement data are in three cross-planes close to the propeller plane, which is located at $x = 0.9846L_{pp}$. On pages 34–47 of [4], further information is found about the flow measurements, including a comparison with measurements at OU and TUHH[3] of the JBC at a different Reynolds number or in a different configuration (double-body model).

The mean axial velocity is compared (simulation-measurements) in figure 5. The agreement is overall relatively good. It is noted that the simulations did not contain any special mesh refinement region around the propeller so it is not specifically designed to provide an accurate flow prediction in the propeller region. The main discrepancy is that the simulations appear to under-predict the development of the symmtrical "hook-shaped" vortex pair. It is still present in the simulations, but not as strong as in the measurements. This is seen, for instance, in plane S2 where the minimal measured value in the hook is $u \approx 0.2V_0$, whereas this corresponding value for the simulation is $u \approx 0.4V_0$ and, furthermore, the simulation does not contain a closed iso-line of u/V_0 in this location. Apart from this, the extent of the low-velocity region is quite similar between simulation and experiments and its qualitative development (in planes S2, S4 and S7) is very similar.

Simulation results, for the prediction of the turbulent kinetic energy, k, in planes S4 and S7, are shown in figure 5. No comparison is made with measurements since data for k is not freely available at the Tokyo workshop homepage. Measurement data for k were however recently published for planes S4 and S7, see pages 41 and 44 of [4], where the large uncertainties in

[3]TUHH=Technische Universität Hamburg

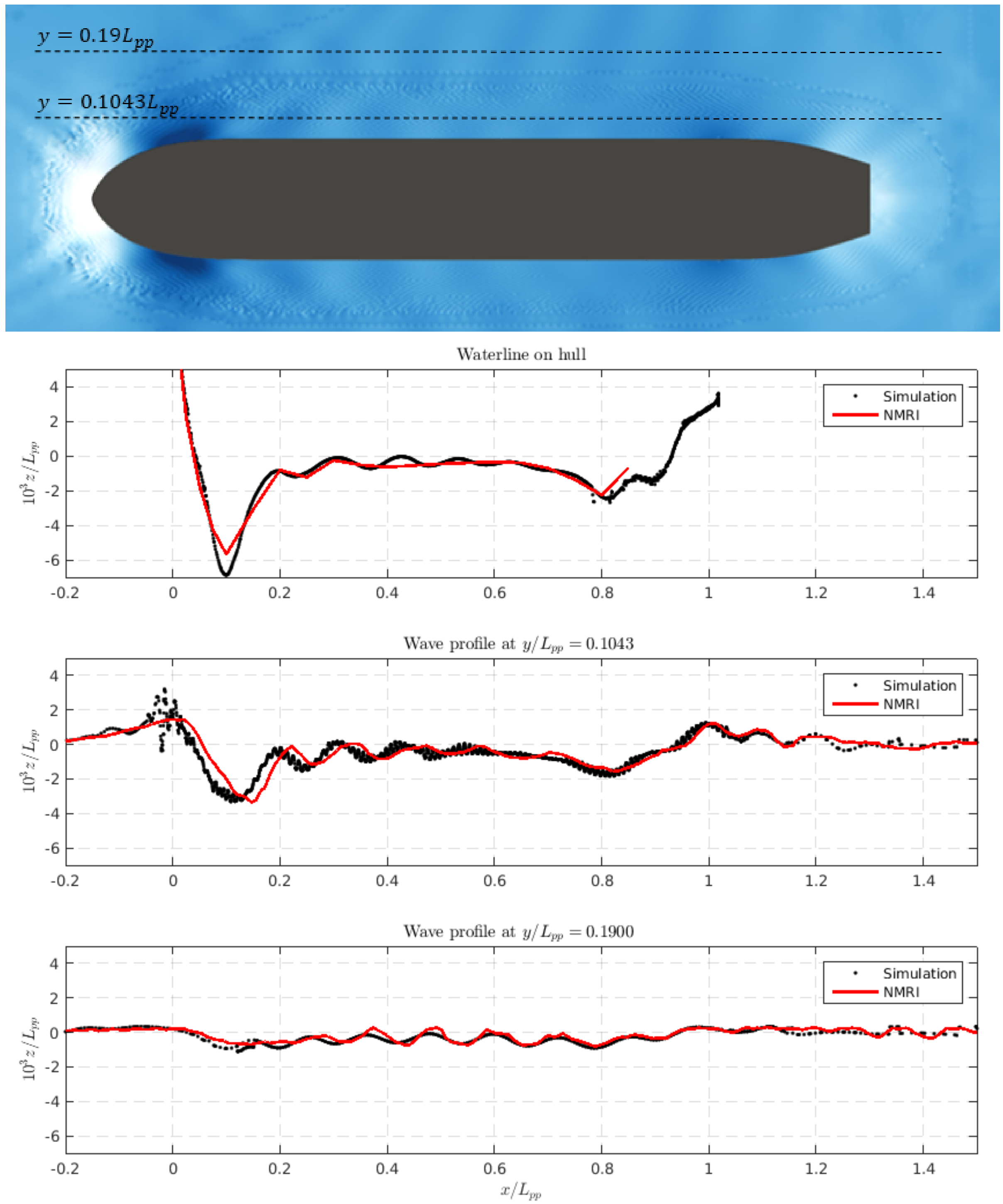

Figure 4: Waterline on the hull and two wave profiles parallel to the hull. Comparison of simulations and measurements, [3]. The top plot illustrates the location relative to the hull, where the wave profiles are taken. The waterline and the two wave profiles are included in one line plot each as indicated in the title to each graph.

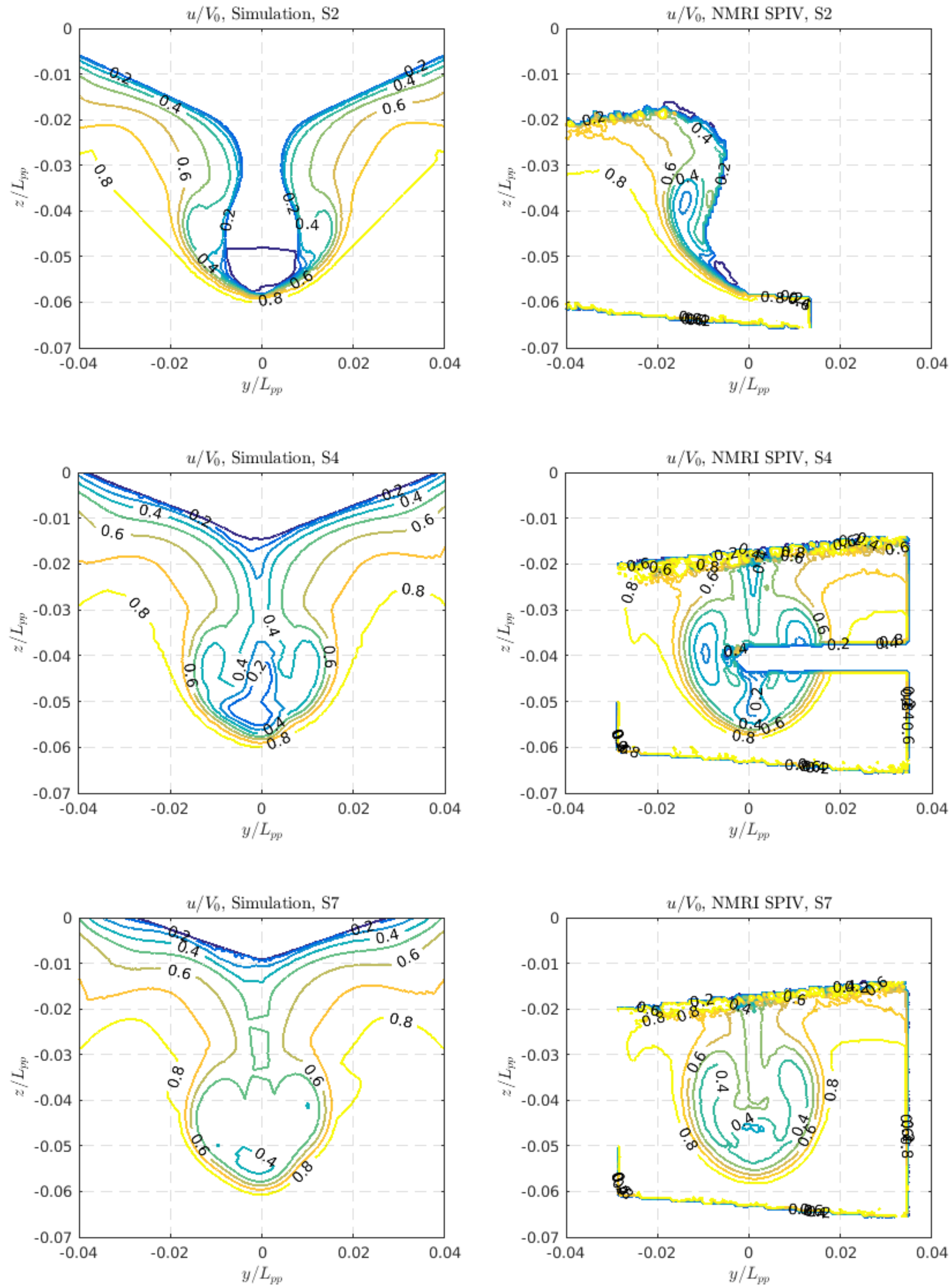

Figure 5: Normalized mean axial velocity, u/V_0, in cross-planes at the stern, $x = 0.9625L_{pp}$ (S2), $x = 0.9843L_{pp}$ (S4), and $x = 1.0000L_{pp}$ (S7), respectively. Simulation results in the left column of plots and SPIV-measurements, [4], in the right column.

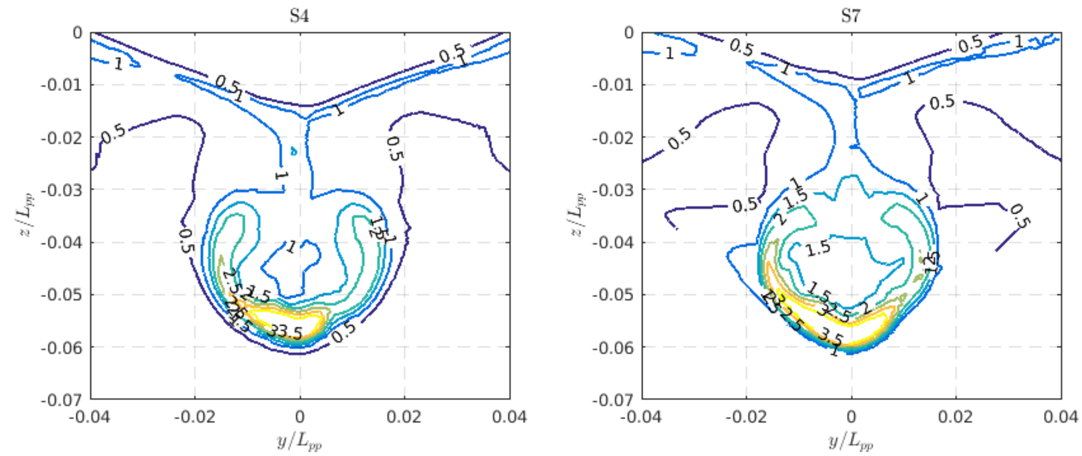

Figure 6: Contour plots of simulation results for the normalized turbulent kinetic energy, $100k/(V_0^2/2)$. Plots in the cross-planes S4 and S7 (see caption of figure 5) at the stern.

the measurements are discussed and, furthermore, the large observed discrepancies between k-measurements in different facilities. Nevertheless, for completeness, the k-plots for the simulation are included here. The results exhibit the expected qualitative behavior, with high k-values in the hook and the peak values in the lower part of it. The predicted peak k-values are at approximately 4%.

In summary, several aspects of the predicted flow field, the ship position and the forces have been compared with measurement data. The ship equilibrium position shows very good agreement with the measured values. The predicted total force, C_t, is 11% higher than that measured which is a disappointing result considering the conclusion in the evalution of the Tokyo workshop CFD results (pp 140 of [4]) show that almost all simulations have this comparison error below 10%, most of them below 5%. The discrepancy in C_r, see table 2, indicates that an inaccurate flow prediction of the stern flow, which then would affect the pressure distribution, may be an explanation. On the other hand, the flow in planes S2, S4 and S7 is relatively well captured, which diminishes the probability that the discrepancy in C_r is due to large errors in the predicted stern flow. Furthermore, the wave field prediction is relatively good as well, which also indicates that the pressure distribution around the hull should be accurate. The main conclusion is that there is room to improve on this simulation case, but that it is sufficiently accurate, with respect to all flow features, to be used as a starting point for any research group interested in using OpenFOAM for ship hydrodynamics generally and specifically in further developing accurate simulation cases for the JBC test case.

5 REPRODUCIBILITY AND DATA AVAILABILITY

An important objective of this paper is that the results should be fully reproducible by other research groups. This is achieved by using an open-source software (OpenFOAM) for the simulations, and making a complete simulation case readily available for free download. All case files necessary to carry out the simulation again should thus be included, and not only post-processed results. In this section, practical aspects of this are discussed, and the approach which is used by the authors to make the JBC case at Fr = 0.142 available is described. Note that the coordinate system used for results presentations in the present paper matches that used for presenting measurements, [3, 4], but in the simulation case a differente cartesian coordinate system was employed.

5.1 OpenFOAM case files made available

There are essentially two possible approaches for making an OpenFOAM case available so that it can be re-simulated by others. The first approach is to only provide minimal initial data. Then the mesh is not provided, but only input data to `snappyHexMesh`, which then must be re-run before the simulation. Likewise, the initialized α-field is not provided, but only input to the utility `setFields` which creates the initial α-field representing a horizontal water surface. Thus, all initial fields contain a constant value specified by a single number in the file which, hence, is quite small.

The second approach is to provide data so that it can be re-simulated, but also to provide the results so that it is not necessary to re-simulate. Here, the complete grid is provided and the initial data is ready for simulation start without additional pre-processing. In addition, the final time step is included, so that no simulation is necessary but results can be verified, and further results extracted directly. These advantages come at the cost of a significantly larger data set.

For the JBC-case presented in this paper, the total data size of the second approach is about 7 GB, without any file compression. The mesh (`polyMesh` directory) has the size 1.8 GB, and the final time step has the size 3.4 GB. The logfile, containing all time step and iteration information, as well as motion time history, is as large as 1.7 GB for the relatively long simulation which covers more than 300 s of physical time. The size of the remaining files is negligible in comparison. E.g. the initial data has the size 19 MB. For the first approach, the largest file is the relatively fine STL-file (hull geometry description) which is input to `snappyHexMesh` and has the size 170 MB. For this particular case, the gain in data size is approximately a factor of 35 in using the first approach, as compared to the second.

The second approach was seleted for the JBC-case at Fr = 0.142, and a complete simulation case was made available, as further described in the next section. One reason for the choice is that the simulation is relatively expensive, ≈80 000 core hours, due to the long simulation time interval. Hence it is practical to have the final time step directly available without re-simulation. Another reason is that the algorithms may not be completely deterministic. In the mesh generation there may be a random element in adjusting grid points in problematic regions. Furthermore, parallel simulation on different decomposition gives slight numerical differences. These differences do not affect the general predictive accuracy of the results, but they may inhibit exact reproducibility.

5.2 Miscellaneous practical aspects

The simulation case data is shared using the easily accessible repository system Figshare[4] under the creative commons license "CC BY 4.0". The dataset was uploaded to the Figshare

[4]https://figshare.com

account of the first author of this paper. The OpenFOAM software is available under the GNU General Public License. The availability of the data is of course not ensured in the same archival manner as e.g. journal publications, but it is expected that it will still be easily accessible in the foreseeable future.

To reproduce the results in this paper it is thus only necessary to install OpenFOAM and to download the JBC-simulation case (Fr = 0.142) from Figshare. The complete case is stored as a compressed tar-archive with the size 1.7 GB. Additional functionality may be needed to post-process the data set, such as e.g. the open-source visualization Paraview[5] or comparable tools.

6 CONCLUSIONS

The paper describes a simulation of the JBC validation case, using the open source software tools of OpenFOAM. Simulation results were compared with measurements concerning ship position, water surface deformation, hull forces and the flow field in the stern. The agreement of the results is reasonable, see section 4 for a full discussion, and the simulation case may be used as a starting point for any research group interested in using OpenFOAM for ship hydrodynamics. In order to make the results fully reproducible, the data files are made freely available on the easily accessible repository system Figshare as explained in section 5.

REFERENCES

[1] Performance, propulsion 1978 ITTC performance prediction method. ITTC - Recommended procedures, 1978.

[2] S. S. Deshpande, L. Anumolu, and M. F. Trujillo. Evaluating the performance of the two-phase flow solver interFoam. *Comput. Sci. Disc.*, 5(014016):1–36, 2012.

[3] T. Hino, N. Hirata, K. Ohashi, Y. Toda, T. Zhu, K. Makino, M. Takai, M. Nishigaki, K. Kimura, M. Anda, and S. Shingo. Hull form design and flow measurements of a bulk carrier with an energy-saving device for CFD validations. In *PRADS2016*, Copenhagen, Denmark, 2016.

[4] T. Hino, F. Stern, L. Larsson, M. Visonneau, N. Hirata, and J. Kim, editors. *Numerical ship hydrodynamics. An assessment of the Tokyo 2015 workshop.* Number 94 in Lecture notes in applied and computational mechanics. Springer, 2020.

[5] C. W. Hirt and B. D. Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comp. Phys.*, 39:201–225, 1981.

[6] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8):1598–1605, 1994.

[5] https://www.paraview.org