Enhanced coupling of solid body motion and fluid flow in finite volume framework

Inno Gatin\textsuperscript{a,}\textsuperscript{*}, Vuko Vukčević\textsuperscript{a}, Hrvoje Jasak\textsuperscript{a,b}, Henrik Rusche\textsuperscript{b}

\textsuperscript{a} University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, Zagreb, Croatia
\textsuperscript{b} Wikki Ltd, 459 Southbank House, SE1 7SJ London, United Kingdom

ARTICLE INFO

Keywords:
6–DOF rigid body motion
Motion–fluid coupling
Seakeeping
Naval hydrodynamics
Foam–extend

ABSTRACT

An enhanced coupling strategy for the resolution of 6–degrees–of–freedom rigid body motion and unsteady incompressible fluid flow in Finite Volume collocated arrangement using PIMPLE algorithm and rigid mesh motion is presented in this paper. The improved coupling is achieved by calculating the 6–degrees–of–freedom motion equations after each pressure correction step in the pressure–velocity PISO (Pressure Implicit With Splitting of Operators) algorithm. Solving the 6–degrees–of–freedom equations after each solution of the pressure equation of the PISO loop accelerates the convergence, leading to smaller number of nonlinear pressure–velocity iterations needed per time–step. The novel approach is verified and validated on a heaving decay case, while the achieved acceleration in terms of the number of PISO loops is demonstrated on seakeeping simulations of a container ship.

1. Introduction

Numerical simulations using Computational Fluid Dynamics (CFD) are frequently used in computational naval hydrodynamics for assessing wave induced loads and motions (Larsson et al., 2013, 2015a, 2015b). There are numerous reasons why wave induced loads and motions of floating objects are important in marine engineering. Fuel consumption of ships sailing in waves is one of them, due to the increase of oil price during the last couple of decades, as well as the increasingly rigorous regulations regarding harmful gas emission. Seakeeping characteristics of ships are important for safety and comfort of crew and passengers, as well as for assessing acceleration loads (e.g. heavy deck equipment, superstructures etc.).

CFD is proving to be a useful tool in predicting behaviour of ships in waves. Numerous publications (e.g. Orihara and Miyata, 2003; Carrica et al., 2008, 2011, 2012; Bhushan et al., 2009; Kim, 2011; Castiglione et al., 2011; Wu et al., 2011; Guo et al., 2012; Miyata et al., 2014; Mousaviraad et al., 2015; Sadat-Hosseini et al., 2013; Simonsen et al., 2013; Tezdogan et al., 2015) tend to depict the accuracy and potential of CFD for solving such problems, using different ways to couple 6–degrees–of–freedom (6–DOF) motion and fluid flow. The coupling of body motion and fluid flow is commonly performed on the level of the nonlinear pressure–velocity loop (SIMPLE or PIMPLE), i.e. after the flow solution rigid body motion equations are solved and the computational grid is moved accordingly. The procedure is then repeated within each time–step until convergence. This is the conventional strongly coupled approach, hereinafter referred to as conventional approach. The PIMPLE algorithm is comprised of multiple PISO pressure–velocity loops, where pressure is updated multiple times per one momentum equation–update (Issa, 1986).

The above mentioned, conventional approach has been verified in numerous publications. Orihara and Miyata (2003) use a predictor–corrector algorithm for the in–house code WISDM–X, where they recalculate the entire flow field after every body motion correction. Castiglione et al. (2011) imply that the in–house code CFDShip-Iowa uses a similar approach, where the complete fluid flow solution is obtained in each body motion–fluid flow iteration. Wu et al. (2011) describe the execution sequence of the CFD code used in their study where a similar procedure is employed. To achieve convergence of the coupling, multiple body motion–fluid flow iterations are needed. Simonsen et al. (2013) and Vukčević and Jasak (2015) reported that a minimum of five pressure–velocity (PISO) loops were needed per time–step to ensure convergence. For the fluid flow itself to converge, smaller number of PISO loops is sufficient, typically two for wave related problems. Hence, the body motion–fluid flow coupling presents a considerable overhead in terms of CPU time.

A modified approach for coupling the rigid body motion equations and fluid flow is described, verified and validated in this paper. Pressure field and body motion are tightly coupled at the body boundary in large scale naval hydrodynamics problems. Pressure

\* Corresponding author.

E-mail addresses: inno.gatin@fsb.hr (I. Gatin), vuko.vukcevic@fsb.hr (V. Vukčević), h.jasak@wikki.co.uk, hrvoje.jasak@fsb.hr (H. Jasak), h.rusche@wikki.co.uk (H. Rusche).

http://dx.doi.org/10.1016/j.oceaneng.2017.08.009
Received 6 December 2016; Received in revised form 13 June 2017; Accepted 7 August 2017
Available online 18 August 2017
0029-8018/ © 2017 Elsevier Ltd. All rights reserved.
influences the body motion through the force acting on the body, while the moving body influences the flow through the change of body boundary velocity and relative grid motion fluxes (see Demirdzić and Perić, 1988). Rigid grid motion is used, i.e. the grid is not deformed when the body is moving. Enhancing the coupling between the pressure equation and the body motion equations has been proposed before for resolving the coupling of fluid flow and elastic bodies (Fernández et al., 2005), however no similar approach is encountered for a special case of rigid bodies that are modelled as boundaries of the fluid domain, where no volume discretisation of the body is present.

In this work the convergence of the body motion–fluid flow coupling is accelerated by further resolving the coupling via updated 6–DOF solutions after each pressure correction equation within the PISO loop in addition to the standard motion update after each PISO loop. The grid position is not updated between every pressure correction in order to save CPU time. This is allowed since relative flux caused by the grid motion does not influence the pressure equation due to its elliptic nature for incompressible flows. Furthermore, the influence of the new grid position is considered negligible since the motions are generally small within a time–step, even for large overall motions (e.g. manoeuvres). We stress that the grid motion and relative fluxes are updated after each PISO loop in a given time–step, correctly accounting for the complete 6–DOF–fluid flow coupling. Tighter coupling leads to a smaller number of PISO iterations needed to ensure body motion–fluid flow coupling convergence, which in turn reduces the overall CPU time.

The benefit of the presented approach over the conventional approach is the tighter coupling of the pressure equation and the 6–DOF equations which dictate the motion of the body, which in turn represents the boundary of the fluid domain. In the conventional approach, the 6–DOF equations are solved once per PISO loop, i.e. once per pressure–velocity coupling. In the proposed approach, 6–DOF equations are solved significantly larger number of times: in addition to the standard update in every pressure–velocity coupling loop, the 6–DOF equations are additionally solved every time the pressure equation is solved.

This paper is organised as follows. First, the numerical model of the enhanced approach for fluid flow–6–DOF coupling is described, comprising the governing equations, brief description of the numerical procedure and a detailed procedure of the novel algorithm. Second, a test case of a heaving cylinder is presented to verify and validate the novel approach by comparing the results with experimental and analytical results. Finally, container ship seakeeping test cases are presented to demonstrate the improvement of convergence of rigid body motion–fluid flow coupling achieved with the new approach, accompanied by a discussion of the results. Finally a brief conclusion is given.

2. Numerical method

The enhanced 6–DOF–fluid flow coupling scheme is implemented in foam–extend (Jasak, 2009), a community driven fork of OpenFOAM open source software, which uses second–order accurate finite volume spatial discretisation with arbitrary polyhedral grid support (Jasak and Gosman, 2001). In this section a brief overview of the discretised governing equations for incompressible two–phase flow is given. The numerical procedure based on the PISO algorithm including the solution of 6–DOF rigid body motion equations is shown. Finally, the novel approach for coupling 6–DOF body motion equations with the pressure equation is presented.

2.1. Fluid flow governing equations

In free surface hydrodynamic problems, the incompressible two–phase flow is governed by the momentum equation, continuity equation and the free surface transport equation. Two phases are modelled with a single set of governing equations, where the discontinuity in pressure gradient and density at the interface is resolved using the Ghost Fluid Method (GFM) (Vukčević, 2016; Vukčević et al., 2017). The GFM imposes pressure jump conditions at the free surface ensuring a sharp transition of fluid properties. For incompressible fluids the conservation of mass is governed by:

$$\nabla \cdot \mathbf{u} = 0, \quad (1)$$

where \( \mathbf{u} \) represents a continuous velocity field in the global coordinate system. For a moving computational grid the momentum equation reads:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \mathbf{u}) - \nabla p = -\frac{1}{\rho} \mathbf{g} - \frac{1}{\rho} \nabla \mu, \quad (2)$$

where \( \mathbf{u} \) is the relative grid motion velocity which stems from the Space Conservation Law (Demirdzić and Perić, 1988); \( \mathbf{r} \) is the effective kinematic viscosity comprising appropriate phase viscosity and turbulent eddy viscosity; \( \rho \) is the density field, and \( p \) stands for dynamic pressure: \( p = \rho - \rho g \cdot \mathbf{x} \). Note that due to the GFM, volumetric fluxes are used for conversion instead of mass fluxes (see Vukčević et al., 2017 for details). Algebraic Volume of Fluid (VOF) (Rasche, 2002) method is used for interface capturing with additional convective term for interface compression:

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

where \( \alpha \) is the volume fraction, and \( \mathbf{u} \) stands for artificial compressive velocity field which is oriented in the normal direction towards the free surface (Weller, 2008). The third term is active only near the free surface due to the nonlinear term \( \alpha (1 - \alpha) \). The details on the evaluation of \( \mathbf{u} \) can be found in Rasche (2002).

Detailed discretisation of temporal derivative, convection and diffusion in (2) in integral form can be found in Jasak (1996), while already discretised equations are used in the text below. The semi–discretised momentum equation for each cell reads:

$$\alpha_f \mathbf{u}_f + \sum_j \alpha_{Nj} \mathbf{u}_{Nj} = \mathbf{b} - \frac{1}{\rho} \nabla P_f, \quad (4)$$

where \( \alpha_f \) stands for the diagonal coefficient, \( \alpha_{Nj} \) for the off–diagonal coefficients, and subscripts \( P \) and \( N \) stand for values in the parent cell centre and neighbouring cell centres, respectively. Parent cell is the cell for which the equation is being solved for, and the neighbouring cells are all the cells which share a face with the parent cell (Jasak and Gosman, 2001). \( \sum_f \) is the sum over all neighbouring faces \( f \), and \( \mathbf{b} \) stands for the source term. Following notation proposed by Jasak (1996), (4) can be written as:

$$\mathbf{u}_f = \frac{H(\mathbf{u}_f)}{\alpha_f} - \frac{1}{\rho} \nabla P_f, \quad (5)$$

where \( H(\mathbf{u}_f) \) presents an explicit operator:

$$H(\mathbf{u}_f) = - \sum_j \frac{\alpha_{Nj}}{\alpha_f} \mathbf{u}_{Nj} + \mathbf{b}. \quad (6)$$

The pressure equation is derived by interpolating (5) on cell faces and substituting into the discretised form of (1), yielding:

$$\sum_f \frac{1}{\alpha_f} \left( \frac{1}{\rho} \nabla P_f \right)_f = \sum_j \frac{s_j \cdot \left( H(\mathbf{u}_f) \right)_j}{\alpha_f}, \quad (7)$$

where \( s_j \) stands for surface normal vector, and subscripts \( f \) denote values at face centres. For a detailed derivation of the pressure equation with the GFM, the reader is referred to Vukčević et al. (2017).

2.2. Numerical procedure

The solution of above equations is achieved in a segregated manner in a PIMPLE loop, a combination of SIMPLE and PISO algorithms,
where the PISO loop is repeated multiple times per time step. The pressure velocity coupling is resolved in the PISO manner, while the repetition of PISO multiple times per time step is specific to SIMPLE algorithm. Within the PISO loop, the momentum equation is solved once, while the pressure equation, (7), is solved multiple times. After each pressure correction step, the velocity field is updated explicitly using (5).

2.3. Coupling of fluid flow and 6-DOF body motion equations

In most naval hydrodynamic problems pressure forces acting on a moving body dominate over viscous forces, resulting in a strong coupling of 6-DOF body motion and pressure at the body boundary. The boundary velocity is determined by solving the 6-DOF body motion ODE, where the right hand side represents the excitation force $\mathbf{F}$ and moment $\mathbf{M}$:

$$\mathbf{F}_p = \sum_{bf} \mathbf{S}_{bf} \rho_{bf} \mathbf{I},$$

$$\mathbf{F}_v = \sum_{bf} \rho_{bf} \mathbf{r}_{bf} \mathbf{S}_{bf} \cdot \mathbf{T}^b,$$

$$\mathbf{M}_p = \sum_{bf} \mathbf{r}_{bf} \times \mathbf{S}_{bf} \rho_{bf},$$

$$\mathbf{M}_v = \sum_{bf} \mathbf{r}_{bf} \times (\rho_{bf} \mathbf{r}_{bf} \mathbf{S}_{bf} \cdot \mathbf{T}^b),$$

(10)

where indices $p$ and $v$ denote pressure and viscous parts, respectively. Pressure and viscous forces are calculated in the global inertial reference frame by integrating the pressure and shear stress on the body surface, respectively:

$$\mathbf{F}_p = \sum_{bf} \mathbf{S}_{bf} \rho_{bf} \mathbf{I},$$

$$\mathbf{F}_v = \sum_{bf} \rho_{bf} \mathbf{r}_{bf} \mathbf{S}_{bf} \cdot \mathbf{T}^b,$$

$$\mathbf{M}_p = \sum_{bf} \mathbf{r}_{bf} \times \mathbf{S}_{bf} \rho_{bf},$$

$$\mathbf{M}_v = \sum_{bf} \mathbf{r}_{bf} \times (\rho_{bf} \mathbf{r}_{bf} \mathbf{S}_{bf} \cdot \mathbf{T}^b),$$

(10)

where index $bf$ denotes face at the body boundary, $\mathbf{T}^b$ is the deviatoric part of the stress tensor $\mathbf{T}$ which is twice the symmetric part of the velocity gradient tensor $\nabla \mathbf{u}$, $\mathbf{r}$ is the radius vector of the face centre with respect to the centre of gravity. In the conventional approach used in PIMPLE algorithms, the pressure force $\mathbf{F}_v$ converges in an oscillatory manner in successive PISO loop during one time step due to the elliptic nature of the pressure equation, while the viscous force exhibits smoother convergence. If the pressure forces are dominant, such oscillations deteriorate the convergence rate of body motion–fluid flow coupling.

The pressure equation and the rigid body motion equations are mutually coupled, where the pressure influences the 6-DOF equations through (10), while the body velocity solution influences the pressure equation through (6). The strong coupling between pressure and body boundary velocity is present even without moving the grid.

In this work an enhanced approach for 6-DOF–fluid flow coupling is proposed, in which 6-DOF body motion equations are solved after each pressure correction equation within the PISO loop, allowing tighter coupling with the pressure equation. After the solution of the pressure equation, the forces acting on the body are recalculated and body motion ODEs are solved:

$$\frac{\partial \mathbf{v}}{\partial t} = \frac{\mathbf{F}}{m},$$

$$\frac{\partial \omega}{\partial t} = \mathbf{I}^{-1} \cdot (\mathbf{M} - \omega \times (\mathbf{I} \cdot \omega)).$$

(11)

where $\mathbf{v}$ is the velocity of the centre of mass of the body, $\omega$ is the rotational velocity with respect to the centre of mass, while $\mathbf{I}$ stands for the tensor of inertia of the body. Fifth-order Cash-Karp embedded Runge-Kutta scheme with error control and adjustable time-step size (Press et al., 2002) is used for integration of rigid body motion equations, while no under-relaxation is used.

The change of the body velocity is inserted in the pressure equation through (6) by updating the source term $\mathbf{b}$ for cells adjacent to the body boundary. The source can be decomposed into the contribution from the boundary condition at the moving body and the remaining part of the source:

$$\mathbf{b} = \mathbf{b}_b + \mathbf{b}(\mathbf{u}_f).$$

(12)

where $\mathbf{u}_f$ stands for the velocity of the body boundary faces, $\mathbf{b}(\mathbf{u}_f)$ is the part of the source term which is a function of the velocity at the body boundary, while $\mathbf{b}_b$ stands for the remaining portion of the source term (other boundary conditions, non–orthogonal correction of the diffusion term etc.). The body boundary velocity is calculated from the rigid body kinematics obtained from (11) as:

$$\mathbf{u}_f = \mathbf{v} + \omega \times \mathbf{r}_f.$$  

(13)

The body boundary, i.e. the grid, is not moved in the pressure loop to save CPU time. Instead, the grid is moved only once per PISO loop, to allow complete coupling with the velocity field and the free surface. Hence the 6-DOF equations are further resolved at the negligible expense of additional rigid body motion ODE solution per each pressure correction equation. The flow chart of the PIMPLE loop including the enhanced 6-DOF body motion coupling is shown in Fig. 1.

3. Decaying heave motion of a cylinder

In this section verification and validation of the proposed approach is presented on a case of 2D decaying heave motion of a cylinder. Verification is performed by refining the spatial and temporal resolution simultaneously, following Eça and Hoekstra (2008). Experimental and theoretical results published by Ito (1977) and Maskell and Ursell (1970) are used for validation, respectively.
The experimental and theoretical data are given in dimensionless form, where the temporal and spatial scales are normalised with the initial displacement \( \delta \) and with \( \sqrt{\delta/R} \), respectively, where \( R \) denotes the cylinder radius. Hence, the cylinder dimensions are selected arbitrarily: \( R = 1 \) m, while the initial displacement is set to \( \delta = 2/3R \) upward. The computational domain is 60 m wide, with a water depth of 20 m and 10 m height above the free surface. Fig. 2 shows the computational domain with boundaries and cylinder position with respect to the free surface. Implicit relaxation zones (Jasak et al., 2015) are positioned at the left and right boundary for wave absorption, and extend 15 m towards the cylinder. Wave velocity boundary condition is set on the left, right and bottom boundary, while zero gradient is set on the top boundary. Zero gradient pressure boundary condition is set on all boundaries except the top, where total pressure is prescribed.

Second order backward temporal discretisation scheme is used, while implicit upwind scheme with deferred second order correction is used to discretise the convection term in the momentum equation. Second order scheme with explicit limited non–orthogonal correction is used for the discretisation of the diffusion term. For the interface capturing equation second order scheme with deferred correction is used for the convection, while turbulence equations are convected with first order upwind scheme. Four PISO correctors are used per time–step for all simulations, where two pressure corrections per PISO loop are employed. All equations are solved to the tolerance of \( \sum (\phi_{i+1} - \phi_i)^2 \leq 10^{-6} \).

3.1. Verification

Verification procedure for unsteady flow from Eça and Hoekstra (2008) is adopted, where four grids are used in order to establish the total uncertainty arising from temporal and spatial discretisation errors. According to the recommendations of Eça and Hoekstra (2014), for complex flow cases at least four refinement levels are needed to get a reliable uncertainty assessment. Refinement ratio \( r = 2 \) is used which defines the ratio between the temporal resolution \( t_{i+1}/t_i \) and spatial resolution \( h_i/h_{i+1} \) of adjacent refinement levels \( i \). Temporal and spatial resolution are refined simultaneously, i.e. \( h_i/h_{i+1} = r \), which is the vertical and horizontal dimension of a typical cell near the free surface; \( r = 0.05 \) s. This results in 19 050, 72 554, 283 386 and 1 120 410 cells for grids corresponding to refinement levels 1, 2, 3, and 4, respectively. The corresponding time–steps are \( t_1 = 0.05 \) s, \( t_2 = 0.025 \) s, \( t_3 = 0.0125 \) s and \( t_4 = 0.00625 \) s. Since the time–step is fixed, the Courant–Friedrichs–Lewy (CFL) number varies between 0.5 and 2.5, with a mean of \( \approx 1.1 \). Since the spatial and temporal resolution are refined simultaneously, the CFL number behaves similarly in all simulations. The computational grid is unstructured, composed mostly of hexahedral cells (\( \approx 95\% \)), and a minor portion of polyhedral cells (\( \approx 5\% \)). The coarsest grid is shown in Fig. 3, where it can be seen that the grid is refined in the vicinity of the free surface.

Eça and Hoekstra (2014) recommend using a least squares approach for discretisation error estimate when the flow field is complex and when unstructured grids are used, which will be summarised here for clarity. Evaluation of \( \delta \)th grid uncertainty for well behaved data sets with four or more refinement levels can be expressed as:

\[
U_i = F_i e_i + \sigma + |\phi_i - \phi_{i\text{obs}}|	ag{14}
\]

where \( U_i \) presents the grid uncertainty for grid \( i \), \( F_i \) is the safety factor, \( e_i \) is the discretisation error, \( \sigma \) is the standard deviation of the least squares fit, \( \phi_i \) is the value of the observed simulation variable on grid \( i \), and \( \phi_{i\text{obs}} \) is the corresponding value obtained from the least squares fit. The data is considered well behaved if the relation \( \sigma < \Delta \) is fulfilled, where \( \Delta \) defines a data range parameter defined as:

\[
\Delta = (\phi_{\text{max}} - \phi_{\text{min}})/(N - 1)
\]

where \( \phi_{\text{max}} \) and \( \phi_{\text{min}} \) denote maximum and minimum value of the variable obtained using different refinement levels. In cases when \( t_i \) and \( h_i \) are simultaneously varied, \( e_i \) is determined as (Eça and Hoekstra, 2008):

\[
e_i = ah_i^p, \quad h_i = (\tau/h_i)^{1/3}
\]

where \( a \) is a constant that needs to be calculated, while \( p \) is the achieved order of accuracy. When more than three refinement level results are available, least squares method is used to obtain \( a \) and \( p \) by minimising the following function:

\[
S(\phi_{\text{obs}}, a, p) = \sum_{i=1}^{N} (\phi_{i\text{obs}} - (\phi_i + ah_i^p))^2
\]

where \( \phi_{\text{obs}} \) presents the estimate of the exact solution. The convergence is considered monotone if \( 0.5 \leq p \leq 2.1 \). The standard deviation of the least squares fit is determined as:

\[
\sigma = \sqrt{\frac{\sum_{i=1}^{N} (\phi_{i\text{obs}} - (\phi_i + ah_i^p))^2}{N - 3}}
\]

Fig. 4 shows the heave signal in time for the four refinement levels, where \( z \) stands for the heave displacement. Fig. 5 shows the enlarged graph in order to see the analysed portion of the signal more clearly. The uncertainty analysis is performed on heave signal characteristics in temporal and frequency domain. In temporal domain, first two extrema of the signal are used, while in the frequency domain the mean value and harmonics with the largest magnitudes are analysed. The harmonic amplitudes are obtained by performing a Fast Fourier Transform on the first 10 s of the signal. Fig. 6 shows the initial residuals of the last solution in the time–step for pressure, momentum and interface capturing equations during the simulation for the finest refinement level. The residuals exhibit similar behaviour for all variables, maintaining at the level of \( O(-5) \) or less during the simulation. Fig. 7 shows the spectrum of the heave signal for all refinement levels, where \( Z \)
stands for the Fourier amplitude of the heave signal. Circles denote the amplitudes that are analysed.

Table 1 shows the complete verification data for two items in temporal domain: \( z_{\text{ext}1} \) and \( z_{\text{ext}2} \), denoting the first and the second extrema of the heave signal; and three items in frequency domain: mean \( Z_0 \), first \( Z_1 \) and second \( Z_2 \) order amplitudes. The uncertainties are calculated for the finest grid, i.e. \( i = 4 \) in (14). Standard deviation \( \sigma \) is smaller than the data range parameter \( \Delta \) for all items, hence the data is well behaved. Since \( 0.5 < p < 2.1 \) for all items, monotone convergence is achieved. Order of convergence is between 1 and 2 for all items except the mean value. Uncertainties are acceptably small for all items ranging from 0.03% to 2.1%, except for the mean of heave motion \( Z_0 \) which exhibits 10% of uncertainty due to its small absolute value.

3.2. Validation

Results obtained using the finest temporal and spatial discretisation are used for the validation including the uncertainties calculated in the previous section according to Eça et al. (2016), where they are compared against experimental data presented by Ito (1977) and theoretical results published by Maskell and Ursell (1970). The comparison is performed for the same items used in the verification study. Fig. 8 shows the comparison of dimensionless heave signals. The present result agrees with theory and experiment reasonably well, within the range set by the theoretical and experimental data. Comparison of temporal and spectral items between the theoretical, experimental and numerical results is presented in Table 2. Relative difference is calculated as \( D_{\phi,\phi'} = (\phi - \phi')/\phi \), where indices denote the corresponding method: \( \phi \) is experimental, \( \phi' \) theoretical and \( \phi'' \) is the result of the present CFD method. The relative differences between the experimental and theoretical results are generally larger than the difference of the CFD result with either of them. It can be noticed that the CFD result corresponds better to the theoretical data, while larger differences occur with respect to the experimental results.

The result comparison including discretisation uncertainties are graphically presented in Fig. 9, where the uncertainties from Table 1 are used. Note that absolute values of corresponding items are shown for better visibility. The error bars, which present the discretisation uncertainty interval, are often very close together and visually form a single line. Generally the numerical results agree better with theoretical
results, especially for the temporal items, \(z_{\text{ext}1}\) and \(z_{\text{ext}2}\).

### 4. Performance comparison on a seakeeping case

In this section a typical seakeeping case is considered in order to compare the enhanced approach with the conventional approach. The two methods are compared in terms of performance, including the 6–DOF–fluid flow coupling sensitivity study, where the number of PISO correctors per time–step is varied.

Seakeeping simulations of KRISO Container Ship (KCS) are performed to assess the acceleration achieved with the enhanced approach for 6–DOF–fluid flow coupling. Regular head waves are imposed at design Froude number \(Fr = 0.261\), while the wave parameters are chosen to correspond to 2.10 CS case from the Tokyo Workshop on CFD in Ship Hydrodynamics (2015) (National Maritime Research Institute, 2015).

In order to compare the two coupling strategies, all parameters of the simulations are the same in all cases except the number of PISO correctors per time–step. Simulations with 2, 4, 8, 10 and 14 PISO correctors are conducted with the conventional approach, while 2, 3, 4, 8 and 14 PISO correctors are used with the proposed approach. Four pressure correction steps are used per PISO corrector in all simulations. \(k\)–\(e\) SST turbulence model is used (Menter, 1994) with wall functions, where the average \(y^+\) on the hull is 50. Mean, first order and phase shift of the first order of resistance, heave and pitch are compared. Moving window Fourier transform for every encounter period is used to obtain frequency domain signals throughout successive periods. Fig. 10 shows an example of the moving window Fourier transform of the first order amplitude of heave. The periodic uncertainty is below 2\% for all simulations, hence it is considered not to have a significant influence on the comparison below. More details on how the periodic convergence is attained can be found in Vukčević and Jasak (2015). The required CPU time per time step is also compared. A cluster with distributed memory is used with nodes: CPU2x Intel Xeon E5-2637 v3 4core, 3.5 GHz, 15 MB L3 Cache, DDR4 2133, with Infini–Band communication. Two nodes, i.e. eight cores are used for each simulation.

KCS model characteristics can be found at Tokyo 2015 CFD Workshop website (National Maritime Research Institute, 2015). Heave and pitch motions are free, and half of the model is simulated using a symmetry boundary condition at the vertical centre plane. Inlet boundary is positioned at \(1L_{PP}\) from the fore perpendicular, outlet at \(2L_{PP}\) from the aft perpendicular, while the farfield boundary at the side of the ship is set at the distance of \(1.5L_{PP}\) from the symmetry plane. The depth of the domain is set to \(1.5L_{PP}\), while the top boundary is set to \(1L_{PP}\) from the free surface. The velocity at the inlet, outlet, bottom and farfield is prescribed from the stream function wave theory, while the dynamic pressure boundary condition is set to zero gradient. Fixed pressure is set at the top boundary, with zero gradient on the velocity field. Wave height to cell height ratio is \(H/\Delta z \approx 20\), while wave length to cell length ratio near the hull is \(\lambda/\Delta x \approx 350\). Implicit relaxation zones (Jasak et al., 2015) are used in order to gradually impose the incident wave field into the computational domain and to damp the waves at the outlet boundary, in order to prevent reflection. Length of relaxation zones are \(\lambda/2\) at the inlet and side boundary, and \(\lambda\) at the outlet. Fig. 11 shows the discretised hull surface, symmetry plane and a plane normal to the longitudinal direction on the main cross section. The rudder is fixed at zero angle, and the model is towed at constant carriage velocity corresponding to Froude number of 0.261. Same grid with 950,000 cells is used in all simulations, while the time step is set to \(\Delta t = 0.0046\) s, which corresponds to 400 time–steps (Vukčević, 2016) per encounter wave period. At least 10 encounter periods are simulated.
in all simulations. Fig. 12 shows one time instance from the simulation, where the incident, diffracted and radiated wave systems can be observed.

The results are compared in Fig. 13. The solution obtained using 14 PISO correctors with conventional approach is used as a reference (denoted by symbol ◦). The conventional scheme result using two PISO correctors is far from the reference (converged) solution for all measured items. With the enhanced approach, two PISO correctors generally produce the solution which is close to the converged solution. It can also be seen that with the conventional approach at least four correctors are needed to obtain acceptable accuracy. In the following text results shall be analysed item by item.

In Fig. 13 the first row of graphs shows the comparison for total resistance $R_T$. For the zeroth order $R_{T0}$ and first order $R_{T1}$ of total resistance, the result obtained with the conventional approach using two PISO correctors is far from the converged solution. The enhanced approach achieves the solution which is close to the converged solution with fewer number of PISO correctors, while the solution does not change considerably with increasing number of PISO correctors, especially for the first order of total resistance.

The second row of graphs in Fig. 13 shows heave results. First order $z_1$ and phase shift $\gamma_{z1}$ of heave confirm that two correctors with the enhanced approach produce the solution close to the reference solution. For the zeroth order $z_0$, the two methods exhibit different behaviour: the conventional approach diverges, while the enhanced approach converges after 8 PISO correctors. It should be noted that the first order and phase shift are more important for motions from a practical point of view compared to mean value, and that mean values of heave and pitch have significantly smaller absolute values.

The third row of graphs in Fig. 13 presents the result comparison for pitch motion. All three items show that the enhanced approach produces a solution close to the converged one with two PISO
correctors, and that the result shows low dependence on the number of PISO correctors.

The main objective of this work is to reduce the required CPU time for transient simulations with body motion. Hence, a comparison of average CPU time per time–step for all simulations is shown in the first graph in Fig. 14. It can be noticed that two correctors with the enhanced approach require slightly more CPU time than two correctors with the conventional approach. This slight difference is caused by solving the motion equations once per each pressure correction equation. However, since smaller number of PISO correctors can be used with the enhanced approach (e.g. 2 instead of 4–8), a savings in CPU time of a factor of two and more can be achieved. Table 3 shows the total CPU time required for individual simulations.

As can be seen in Fig. 14, a different trend of the number of pressure equation linear solver iterations is exhibited by the two approaches, where the enhanced approach uses fewer number of iterations for small number of PISO correctors, however the conventional approach seems to converge to a smaller number of iterations. Nonetheless, the difference in the number of iterations is not significant in both cases. Krylov subspace Conjugate Gradient (CG) linear system solver with Cholesky preconditioner has been used in all simulations for the pressure equation.

Fig. 15 shows the comparison of the \( L_1 \) norm residuals of the last PISO corrector in the time–step, for velocity (4) and pressure (7) equations, averaged over all time–steps in the simulation. The residuals are shown with respect to the total number of PISO correctors used in the simulation. The velocity residuals represent the average of the component–wise time–step averaged residuals. For small number of PISO correctors, the residuals are lower for the enhanced approach, while both approaches converge to a similar value of \( \approx 5\cdot10^{-6} \) for pressure and \( \approx 3\cdot10^{-6} \) for velocity. The residuals of the 6–DOF motion equations are not shown here since they are few orders of magnitudes smaller than the fluid flow equations residuals, hence they are not relevant for the comparison.

4.1. Time–step sensitivity study

In order to determine the sensitivity of the results with respect to the number of time–steps per encounter wave period, a temporal resolution refinement study is performed. The sensitivity is investigated by using 25, 50, 100, 200, 400 and 800 time–steps per encounter wave period. Six PISO correctors and the coarse 600,000 cells grid is used in all simulations.

Fig. 16 shows heave, pitch and resistance amplitudes with respect to the number of time–steps per encounter period. All items converge with increasing temporal resolution, while first order amplitudes show smaller sensitivity then the mean values with respect to the temporal resolution. Hence, significant savings in computational time can be achieved by using coarse temporal resolution, while attaining reasonable accuracy. The difference in computational time between 25 and 800 time–steps per encounter period is 32 times. Fig. 17 shows the phases with respect to the number of time–steps per one encounter wave period. The phases are more sensitive to the temporal resolution, indicating that the dispersion error is higher than the dissipation error. Since phases are generally less important, especially in the early design stages, coarse temporal resolution presents an attractive option for quick estimation of seakeeping performance.

<table>
<thead>
<tr>
<th>N</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>8</th>
<th>10</th>
<th>14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enhanced</td>
<td>19.0 h</td>
<td>11.5 h</td>
<td>34.6 h</td>
<td>68.0 h</td>
<td>N/A</td>
<td>112.5 h</td>
</tr>
<tr>
<td>Conventional</td>
<td>16.5 h</td>
<td>N/A</td>
<td>32.0 h</td>
<td>62.0 h</td>
<td>83.0 h</td>
<td>113.0 h</td>
</tr>
</tbody>
</table>

Table 3. Total CPU time for 20 encounter periods for individual seakeeping simulations.
An enhanced method for coupling body motion and fluid flow in CFD simulations is presented, verified and validated in this paper. Along with the usual update of 6-DOF body motion equations after the converged pressure–velocity–free surface solution, 6-DOF motion equations are additionally solved after each pressure correction equation in the PISO loop, without moving the grid. The method accelerates the convergence of body motion–fluid flow coupling, reducing the number of PISO correctors per time–step while preserving the accuracy.

The present approach for coupling body motion and fluid flow is verified and validated on a heave decay simulation of a cylinder. The verification showed that the achieved order of accuracy is in accordance with the employed discretisation schemes, verifying the present method. The order of accuracy ranges from 0.89 to 1.93, while the
uncertainties are reasonably small. Comparison with experimental and theoretical results of heave motion verified that accurate results are obtained, where the numerical results fall in the range between the experimental and theoretical data for majority of the quantities.

To test the acceleration which can be achieved using the enhanced scheme, simulations of a KCS model in regular head waves are performed using the conventional approach and the enhanced approach with varying number of PISO correctors. Mean, first order and phase shift of resistance, heave and pitch motions are compared.

The analysis of results showed that the proposed enhanced 6–DOF–fluid flow coupling approach takes less PISO correctors to reach convergence, whereas conventional approach demands four to eight PISO correctors to achieve satisfactory accurate solution. With the enhanced approach, two PISO correctors are sufficient to achieve satisfactory results for mean of resistance and first order amplitudes of motion, which are industrially relevant. Also, accurate prediction of phase shifts is obtained. Hence, significant savings of a factor of two and more in terms of CPU time can be achieved using the proposed approach, while retaining accuracy.

For future research, the issue of added mass instability will be addressed to investigate weather the proposed method enhances the stability of simulation in case of numerically unstable cases.

Acknowledgements

This research was sponsored by Bureau Veritas under the administration of Dr. Šime Malenica and Dr. Quentin Derbanne.

References


