Hydroelastic analysis of a flexible barge in regular waves using coupled CFD-FEM modelling

P.Lakshmyarayanana, P.Temarel & Z.Chen
Fluid Structure Interactions Group, University of Southampton, Southampton, U.K

ABSTRACT: The aim of this paper is to investigate the wave-body interaction of flexible floating bodies by coupling RANS/CFD and Finite Element software. A combination of overset and morphing approaches and finite volume solution to allow for the motion of a barge at the free surface is used. Results are presented for the motion response of the three-dimensional (3-D) barge, treated both as rigid and flexible body, in regular head waves using STAR-CCM+, the latter carried out by a two-way coupling between Star-CCM+ and Abaqus. To illustrate this application, the structure of the flexible barge is modelled as a beam, in line with the flexible backbone model used in experiments. The RAOs of vertical displacements, at a number of positions along the barge, calculated using this coupling technique are compared against experimental measurements and two-dimensional (2-D) linear hydroelasticity predictions.

1 INTRODUCTION

Modern seakeeping computations of ships are carried out using a variety of techniques ranging from two-dimensional (2-D) strip theory using potential flow methods to solving fully nonlinear unsteady RANS (Reynolds-averaged Navier-Stokes equations) methods. Application of CFD (Computational Fluid Dynamics) to study wave-body interactions of ships and offshore structures using RANS have increased over the years due to increase in computational power (ISSC. 2012). Traditionally, seakeeping analysis of ships is carried out treating them as rigid bodies. However, the ever increasing size of ships and offshore platforms have resulted in ‘softer’ or flexible hulls which require hydroelastic effects to be taken into account when predicting fluid-structure interactions (Bishop & Price 1979). Such investigations are predominantly experimental, using flexible backbone models, or numerically using potential flow solvers (Bishop et al. 1977). Although partial nonlinear potential flow methods are also used, RANS/CFD can fully take into account the free-surface and body nonlinearities as well as viscous effects, making it more efficient and realistic for some problems (Brizzolara et al. 2008).

Presently the majority of investigations using RANS/CFD and FEA are carried out using one-way coupling. When the deformations of the structure are large enough to significantly affect the flow field around it, one-way coupling would omit some important fluid-structure interactions. A two-way coupling method will be more suitable in such cases. Few investigations employing two-way coupling have shown promising results (Paik et al. 2009) (Kim & Kim 2009), but need further investigations. In this study a two-way coupling between a finite volume CFD method, using Star-CCM+ (version 8.04), and a finite element method (FEM), using Abaqus (version 6.13-1), is applied to assess the hydroelasticity of a flexible barge in regular head waves. Only symmetric distortions of the barge for a number of wave frequencies are employed and compared against experimental measurements (Remy et al. 2006). 3-D computations are first carried out treating the barge as a rigid body to establish the influence of domain size and mesh refinement along the free surface and the barge. A two-way coupling is then established between Star-CCM+ and Abaqus to investigate the hydroelastic response. The coupling takes place through exchanging pressures and displacements between Star-CCM+ and Abaqus more than once every time-step, namely implicit scheme. The structure is modelled as a non-uniform Timoshenko beam with properties, such as stiffness and mass distribution, as per the model test data. Numerical predictions are also obtained using 2-D hydroelasticity (Bishop et al. 1977). RAO of vertical displacements at various locations along the barge is compared against experimental measurements and numerical predictions to validate the coupling method used.
2 NUMERICAL METHOD

2.1 Finite Volume Method

The CFD software used for all computations in this paper is Star-CCM+. Here we present only a brief description of the numerical method implemented and a detailed theoretical background is provided by Ferziger & Peric (2003). The numerical method used in Star-CCM+ is a finite volume (FV) method in which the flow is assumed to be governed by RANS equations. The RANS equations reduce to the well known Euler equations for the case of inviscid flow. First, the spatial fluid domain is discretized into a finite number of control volumes (CVs) or cells. The integral form of conservation equation, with the initial and boundary conditions, is then applied to cell centers and simplified into an algebraic system of equations. The governing equations not only contain surface and volume integrals but also time and spatial derivatives. They are solved using a segregated iterative algorithm, called SIMPLE. All integrals are computed using midpoint rule. The Hybrid Gauss-Least Square gradient method is used to solve the transport equations.

Free surface flows are implemented using the Volume of Fluid (VOF) tracking method. In order to account for the position of free surface in multiphase flows and allow for its arbitrary deformation, an additional equation is solved for the volume fraction $c$.

When the motion of a body at a free surface is involved, the position of the body is updated at each iteration. The equations of motion of the body are solved to obtain the velocities and, hence, update the displacements and rotations. The fluid grid is adjusted at every outer iteration to follow the updated position of the moving body.

2.2 Grid Adaptation in FSI

In the present study, grid adaptation to follow the motion of body is implemented by two different methods, namely morphing and overset grids, the choice depending on the problem solved. In the case of a two-way coupling, the nodal displacements imported from Abaqus redistribute the mesh vertices by generating an interpolation field throughout the domain. The deformation of the fluid grid must conform to the body and also maintain a good quality of finite volume grid. The arbitrary motion of the mesh vertices is taken into account when solving the fluid transport equations. Star-CCM+ uses a “space conservation law” to balance the volume of a CV as a function of time and the motion of the surface.

Morphing could create problems in the case of large body motions and waves. The deformation of the entire grid could result in the free surface to fall outside the refined region of the grid, resulting in high numerical diffusion before the wave reaches the body. To avoid this problem, an overset or overlapping grid can be used. In this case two regions, background and overset are created, where the background grid is adapted to the free surface. The overset grids are attached to the floating body and move with it freely depending on the motion response.

For rigid body simulations of the barge in head waves, grid adaptation has been carried out using overset grids. In the case of two-way coupling, both morphing and overset is applied to move the grid to follow the barge motions. Morphing condition is set to the boundaries of the barge and it deforms due to the nodal displacements supplied by Abaqus. Floating condition is set as the Morpher boundary condition for the overset, so that it moves freely in accordance with the grid deformation applied by the morpher.

2.3 Coupling Scheme

The coupling schemes control the sequence of data exchanges in the simulations. There are two major coupling schemes in Star-CCM+. For a loosely coupled problem an explicit scheme can be chosen. In this scheme, the data or field exchange takes place once every time step. An implicit scheme is chosen when strong coupling is sought as it is more stable than the former, but at a higher computational cost. In the implicit scheme, field exchanges between the software take place at every single iteration within a time step. The explicit scheme was tested with even very small time steps but resulted in pressure divergence.

2.4 Field data exchange

The coupling is implemented by exchanging pressure and nodal displacements, the so called field data, between Star-CCM+ and Abaqus. The geometry of the floating body must have the same dimensions and coordinates in both software; otherwise the co-simulation will fail due to inconsistency in topology.

The response of the fluid to the structural deformations is expressed through the grid flux term. It represents the ratio of volume swept due to the movement of cell face from one time step to the next time step. Grid flux intensity can be calculated by the product of normal velocity of the face and its area.

In transient simulations, the initial conditions are far from realistic. For the simulation to settle down during the initial phase, it is recommended to relax the grid flux term by lowering the grid flux URF (under relaxation factor). Large fluctuations in pressure at the fluid-structure interface can be decreased using a lower value of grid flux URF. However, a value of URF less than 0.5 will lead to an unrealistic, time-inaccurate solution, especially for problems requiring dynamic accuracy.
2.5 2-D Hydroelasticity analysis

Generalised coordinates for rigid and flexible motions of the barge is calculated using the 2-D hydroelasticity method by Bishop et al. (1977). In brief, the strip theory is used to calculate the hydrodynamic properties of the barge, using Lewis form representation. The barge structure is modelled as a Timoshenko beam. Modal summation is employed to represent vertical displacement, bending moment and shear force at a specified location. The resultant unified equations of motion in regular waves provide the requisite generalized coordinates for a range of wave frequencies.

3 NUMERICAL SIMULATIONS OF A BARGE IN REGULAR WAVES

3.1 Barge Characteristics

The experimental model of a flexible barge consisting of 12 connected caissons is considered for validation of the present numerical method (Remy et al. 2006). Each caisson is clamped to a steel rod which is placed at 57 mm above deck level. The rod has a square cross-section of 1 cm × 1 cm. All the caissons are rectangular sections, except for the bow caisson, which has a bevelled shape. The towing tank dimensions are 30 m × 16 m × 1 m.

The main characteristics of the barge and the flexible rod are given in Table 1.

Vertical, horizontal and torsional bending was allowed for the barge model and tests were conducted in regular and irregular waves of varying headings. In the case of regular wave, the barge motions were measured at 6 different locations, as shown in Table 2 (x, y and z measured from stern, centerline and the keel of the barge) for a number of wave periods.

Table 3: Test Conditions

<table>
<thead>
<tr>
<th>Wave Period (s)</th>
<th>Wave Frequency (rad/s)</th>
<th>Wave Length (m)</th>
<th>Wave Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.8</td>
<td>3.490</td>
<td>5.058</td>
<td>100 mm</td>
</tr>
<tr>
<td>1.6</td>
<td>3.926</td>
<td>3.996</td>
<td>100 mm</td>
</tr>
<tr>
<td>1.2</td>
<td>5.235</td>
<td>2.248</td>
<td>100 mm</td>
</tr>
<tr>
<td>1.0</td>
<td>6.283</td>
<td>1.561</td>
<td>100 mm</td>
</tr>
<tr>
<td>0.9</td>
<td>6.981</td>
<td>1.264</td>
<td>100 mm</td>
</tr>
</tbody>
</table>

3.3 Computational Domain

A 3-D domain is used for all CFD calculations, with x along the barge and y and z in the athwartships and vertical directions, respectively. The lengths of the domain in the inlet-outlet and side wall directions are generally calculated based on \( L_{PP} \) or wave length (\( \lambda \)) based on similar ship-wave interaction studies (Peric et al. 2007) (Seng et al. 2012). In the present study, the wavelength to barge length (\( \lambda/L \)) ratio varies from 2 to 0.4. For \( \lambda/L \geq 1 \), the inlet and wake region is located at about 1.5 \( \lambda \) and 2\( \lambda \), respectively, from the barge. The length of inlet and wake region for cases \( \lambda/L \leq 1 \) is 2.0 \( L_{PP} \) for both. A numerical beach is provided at the outlet to damp the waves and prevent any reflections. The length of this damping region is set to 1.5 \( \lambda \). In the CFD simulations, the length of the side wall (y-direction) is fixed as 8 meters (same as the tank) on one side of the barge for all cases. Initially, a reduced length of the side wall (6 meters) was tested for a few frequencies. They showed evidence of wave reflections from the side walls after 4-5 wave periods. The domain sizes selected for each wave frequency, for both rigid body and coupled simulations, are shown in Table 4. Symmetry condition is used for rigid body simulations, whereas full domain is modelled for the co-simulation cases.
Table 4: Summary of domain sizes against wave lengths

<table>
<thead>
<tr>
<th>Wave Length (m)</th>
<th>Location of Inlet (m)</th>
<th>Wake Region (m)</th>
<th>Damping Zone (m)</th>
<th>Side wall * (m)</th>
<th>Water Depth (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.058</td>
<td>7.6</td>
<td>10</td>
<td>7.6</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td>3.996</td>
<td>6.0</td>
<td>8.0</td>
<td>6.0</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td>2.248</td>
<td>5.0</td>
<td>5.0</td>
<td>3.5</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td>1.561</td>
<td>5.0</td>
<td>5.0</td>
<td>2.5</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td>1.264</td>
<td>5.0</td>
<td>5.0</td>
<td>2.0</td>
<td>8.0</td>
<td>4.0</td>
</tr>
</tbody>
</table>

3.4 Meshing Strategy & Boundary Conditions

Star-CCM+ provides various volume meshing models. In all the present investigations, a combination of trimmer, extruder and overset mesh is used. For the same wave frequency, the mesh refinement used for both rigid body simulations and co-simulations is identical. Trimmer mesher is a robust and efficient method of producing high quality hexahedral meshing with minimum cell skewness. Once the core mesh is created, the extruder mesh produces orthogonal extruded cells for user specified boundaries only. The mesh is extruded from the specific boundary in the normal direction based on the user specified extrusion parameters (i.e. number of layers, stretching ratio and extrusion magnitude). The side wall, in y-direction, and the outlet in all models is extruded using appropriate extrusion parameters. Not only does this aid in saving the global cell count but also dissipates the waves in the far field due to gradual coarsening of grid size. Nevertheless, the mesh growth in the extruder region was kept under 1.1 to prevent any numerical reflections arising due to sudden change in grid sizes between adjacent cells (STAR-CCM+ 2012).

The mesh was refined along the free surface region, near the barge and in the wave radiation zone around the barge using volumetric controls. A typical mesh, corresponding to 1.2s wave period, is shown in Figure 1. Mesh refinement was carried out based on the disturbed wave contour around the body. After testing a few cases it was noted that the waves radiated out of the body in a circular pattern. Hence, refinement around the body was also carried out in a manner so as to capture the disturbed wave pattern. In the free surface region, 45-60 cells are placed per wavelength and 12-15 cells per wave amplitude. Around 320 cells per wavelength and 160 cells per wavelength are clustered in the near body region (bow, stern and around body) and wave radiation zone, respectively. The global mesh count for the co-simulation case varied from 2.3 million to 13.6 million.

Boundary conditions were selected so that they mimic the conditions of a towing tank. At the velocity inlet boundary the kinematics of the wave, i.e. the position of the free surface and velocity of the first-order wave as field functions are prescribed. At the outlet boundary, the outlet pressure and the position of free surface is prescribed. The pressure at the outlet is set to the hydrostatic pressure of the wave. All other boundaries are set to no-slip wall condition.

3.5 Rigid Body Simulations

This section describes the numerical setup and settings specific to the rigid body barge simulations. Due to symmetry of this problem, the computational domain only extends to the port side of the barge.

After an initial orientation of the body is specified by the user, Star-CCM+ automatically creates a new Cartesian coordinate system which is updated showing the position and orientation of the barge throughout the simulation. Body release and ramp time were specified for all simulations which are calculated on the basis of the time step. It is best to allow the fluid flow to initialise and become steady before the calculation of body motions commences. A typical value of 50 time steps is specified as a release time. At the release time, forces and moments are suddenly applied on the body, and can cause shock effect. To minimise this and facilitate a more robust solution by reducing oscillations, a ramp time equal to 10 times the release time is specified.

The VOF under-relaxation factor is decreased from the default value of 0.7 to 0.6. This implies that a fraction of the newly computed solution will be supplied to the old solution, and has been done to increase stability. Consequently, the number of inner iterations was raised to reach a convergent solution. Computations were carried out using inviscid fluid model with an implicit unsteady solver. The 2nd order scheme is chosen for temporal discretisation and convection of segregated flow solver and VOF solver. Time step for each simulation was chosen such that the Courant number on the free surface at all times is less than 0.5.
3.6 Co-Simulation of the barge

This section details the fluid solver settings specific to the coupled simulations. The boundary conditions are those used in the rigid body simulations. Co-simulations were also carried out using an inviscid flow model using implicit unsteady solver. The implicit coupling scheme was chosen for all simulations.

The grid flux URF is lowered from a default value of 1.0 to 0.8, implying that the fluid response to structural response is slightly reduced by a fraction commensurate to the URF. This was done to provide stability as many attempts with a higher grid flux URF resulted in quick pressure divergence. For some cases, where pressure peaks were observed even when the morpher was running smoothly, the URF for pressure was lowered to 0.3 from a default value of 0.4. Similar to the rigid body cases, all simulations were run using 2nd order temporal and convective schemes. When carrying out co-simulations using Star-CCM+ and Abaqus, the FSI boundary has to be defined explicitly. The barge boundary is set as the FSI boundary in the fluid mesh.

3.7 F.E Model

The structural or finite element mesh is modelled in Abaqus. When beam elements are used to represent structural models, they have to be linked to surface elements that define the actual wetted surface of the body. In this study, the flexible barge is represented using a 2-D beam model, with 48 beam elements. All material and geometric properties are modelled in line with model test data, shown in Table 1.

The elements chosen are the 2-node linear beam element B31 and 4-node quadrilateral surface element SFM3D4, the latter representing the barge surface. B31 is a Timoshenko beam element allowing for transverse shear deformation. Abaqus automatically calculates the transverse shear stiffness values required in the formulation of element. Please note for this investigation zero structural damping was used, since the frequency range in experiments and CFD/FE simulations was below the first resonance. Surface elements have no inherent stiffness but may have mass/ unit area, though none is specified in this case. They can be used to transmit only in-plane forces and have no bending or transverse shear stiffness. The dummy surface elements are linked to the nodes on the beam elements using kinematic coupling constraints. A large number of nodes or surfaces can be constrained to the rigid body motion of control nodes (in this case the beam element nodes) using kinematic coupling. All six degrees of freedom are constrained in the kinematic coupling of beam nodes and the dummy surface, in the sense that the beam deformations are imparted on to the barge hull. The total mass of the barge is distributed on the beam elements.

For the 2-D hydroelasticity analysis the barge is represented as a non-uniform beam element divided into 48 sections, to achieve consistency with the FE model. The mass distribution, moment of inertia for each segment, is similar to the FE model. No rotary inertia is specified for the beam elements.

![Figure 2. Finite element mesh with the beam and dummy surface linked using kinematic coupling.](image)

4 RESULTS AND DISCUSSION

4.1 Modal Analysis

Modal analysis was performed in Abaqus using Block Lanczos eigen value extraction method. The natural frequencies and mode shapes obtained in Abaqus were compared against calculations performed using the finite difference method applied to a non-uniform beam (Bishop et al. 1977), to ascertain the accuracy of the modelling. The dry hull natural frequencies for the first 5 flexible modes are shown in Table 5.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Abaqus</th>
<th>2-D hydroelasticity</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-node</td>
<td>6.01</td>
<td>6.03</td>
</tr>
<tr>
<td>3-node</td>
<td>16.43</td>
<td>16.49</td>
</tr>
<tr>
<td>4-node</td>
<td>32.00</td>
<td>31.10</td>
</tr>
<tr>
<td>5-node</td>
<td>52.66</td>
<td>52.73</td>
</tr>
<tr>
<td>6-node</td>
<td>78.42</td>
<td>78.34</td>
</tr>
</tbody>
</table>

4.2 Simulation results

The motion responses of the barge treated as rigid and flexible body in regular head waves are presented in this section. Predictions using the inviscid flow option in Star-CCM+ were compared against experimental measurements by Remy et al. (2006) and numerical calculations using 2-D linear hydroelasticity, the latter providing both rigid and distortional displacements along the barge.

Wave elevations were recorded at one L<sub>pp</sub> in front of the barge. Time history of the wave elevations revealed very little wave dissipation, and maximum decrease in wave height was around 5-6% as the
simulation progressed. Average wave amplitude over 8-10 wave periods was used to calculate the RAOs. The heave and pitch RAOs obtained from the rigid body CFD simulation, denoted by STAR, and the corresponding 2-D hydroelasticity rigid as well as distortional principal coordinate amplitudes, denoted by MARS, are shown in Figure 3.

The rigid body response of the barge, obtained from STAR-CCM+ and denoted by STAR_RIGID, is compared with the 2-D potential flow predictions, denoted by MARS_RIGID, in Figure 4. It can be seen that both numerical results predict the rigid motion response with relatively good accuracy, although discrepancies were noted especially towards the forward end of the barge, e.g. points 1 and 3.

At the forward section of the barge, strong bow waves were seen to develop in the CFD simulations from a wave frequency of 3.926 rad/s and upwards. In addition, with increase in frequency, diffraction becomes more dominant resulting in strong localized bow waves. The instantaneous wave pattern around the rigid barge, for frequencies of 3.926 and 6.0 rad/s, is shown in Figures 5 and 6, respectively. In general, very good agreement is noted at all frequencies at locations from around amidships towards the stern of the barge. At point 12 near the stern of the barge (0.19 m), the RAO of the vertical displacement predicted by CFD at 3.926 rad/s is however larger than the linear 2-D linear potential flow analysis. The instantaneous wave contour plot at this frequency shows strong bow and stern waves influenced by pitch motion, as seen in Figure 5. A peak in pitch response can be seen at around 4 rad/s in Figure 3.

The reason for the discrepancy between CFD and 2-D linear analysis predictions at the forward and stern sections of the barge is mainly due to the influence of strong localized wave systems, not very well predicted using a linear potential flow theory. The predictions using CFD is considered more reliable in this case since it accounts for the nonlinear interactions between wave-body. Strong 3-D effects and the bevel shape of the bow could also be one of the influencing factors for the differences observed in the two numerical results for the rigid body analysis.

RAOs of vertical displacements, for the barge treated as a flexible body, are also shown in Figure 4. Those obtained by 2-D hydroelasticity and the two-way CFD/FEM coupling are denoted as MARS_FLEX and STAR_FLEX, respectively. Very good agreement can be seen between the CFD/FEM coupling method and experimental measurements. Comparisons made between the 2-D hydroelasticity numerical prediction and the CFD/FEM coupling also show fair agreement, with some discrepancies observed at higher frequencies. The reason for these differences is attributed to the strong diffraction effects at higher frequencies also prevalent in the flexible body motions. From the amplitude of the 2-node principal coordinate, shown in Figure 3, resonance appears to occur at around 7.5 rad/s, where computations were unfortunately not carried out.

When the rigid and the flexible body responses are compared with each other, vertical displacement of flexible barge is lower, at lower frequencies, towards the bow and stern part of the barge. A clear difference between the two can be observed in the wave contours at 3.926 rad/s by comparing Figure 5 and Figure 7. Strong bow and stern waves are developed at this frequency for the rigid body motions, whereas the flexible body tends to deform following the wave resulting in weak localized waves. This is the reason for larger vertical displacements in the rigid body simulation when compared to the flexible body. At higher frequencies diffraction effect dominates, which results in a similar wave contour around the body for both rigid and flexible body approaches (see Figures 6 and 8), leading to a small difference in predicted vertical displacements.

There are differences between the coupled CFD/FEM simulation results and the experimental measurements in some cases. In general, the predicted response of the flexible barge from amidships to the forward end agrees better with the corresponding measurements. The RAOs are slightly over predicted towards the aft end. It is thought that these differences can be improved by using more refined grids and fine tuning the numerical parameters used in the coupling.

The difference in mesh resolutions between Star-CCM+ and Abaqus could also be a source of instability. The finite element mesh is much coarser when compared to the Star-CCM+ mesh. Investigations using a finer mesh need to be carried out to study grid convergence.

In the coupled simulations constant co-simulation time of 0.005s is set for all cases. The grid size close to the body is quite small and the deformations of the barge are normal to these thin cells. It creates a scenario where the FSI boundary could possibly move more than the thickness of the cell near the boundary. A combination of expected motion and the thickness of cells near the FSI boundary define the time step and co-simulation time. Possibility of using different time steps and their effect on the solution has to be further investigated.
Figure 4. RAOs of vertical displacements along the barge, point 12 near the stern and point 1 near the bow, for both rigid and flexible body analyses.

Figure 5. Instantaneous wave contour around the rigid body at a 11.5 s wave frequency of 3.926 rad/s.

Figure 6. Instantaneous wave contour around the rigid body at 9.5 wave frequency of 6 rad/s.
Comparisons made between the present coupling technique and the experimental measurements showed very few discrepancies. Very good agreement was observed at relatively low frequencies, but slight differences were noted at higher frequencies. Strong diffracting wave systems are developed at these higher frequencies and they may be influencing the motion at the bow and stern sections. It is thought that the predictions at these relatively high frequencies can be improved by appropriate mesh refinement of fluid and structural models and coupling parameters (such as time step, URF’s etc). This will be studied in detail in future investigations. Of the two hydroelasticity numerical methods, predictions using the coupled CFD method showed a far better agreement with experiments as it allows for nonlinearities. Influence of flexibility is clearly seen in the relatively low frequencies as the body deforms with the wave resulting in weak stern and bow waves, hence; lower vertical displacements when compared to the rigid body approach.

The results show that the coupling technique investigated is reliable and compares well with experimental measurements. The next stage of the investigations will involve applying the coupling technique to predict the forces and bending moments of the barge. Special attention will be focused at higher frequencies where resonance occurs for the 2-node vertical bending mode.

5 CONCLUSIONS

The time domain hydroelastic investigation is carried out using commercially available software Star-CCM+ and Abaqus. The field equations are coupled using an iterative implicit scheme, and a Full Newton solution technique is used for solving the structural dynamics. The numerical solution is compared to experimental measurements and 2-D linear hydroelastic predictions. Calculations are carried out for both rigid body and flexible structural idealisations. Very good agreement is achieved between time domain predictions and experimental measurements in most cases, with some exceptions especially at the forward section of the barge.

Although the comparisons for rigid body motions between the two numerical methods agreed well overall, large differences were observed in the bow and stern regions of the barge at lower frequencies. This is believed to be due to strong bow and stern waves systems mainly influenced by the pitch, which are captured in CFD. CFD solves nonlinear Navier-Stokes equations, even when an inviscid flow model is selected which makes it more realistic than the linear potential flow code. Nevertheless, it should be noted that the rigid body approximation is not suitable for this very flexible barge.

6 REFERENCES