CORA Cork Open Research Archive
Cartlann Taighde Oscailte Chorcaí

Title	Offshore wind farm service vessel, hull design optimisation
Authors	Shanley, Matthew;Murphy, Jimmy;Molloy, Padraig
Publication date	2012-10-01
Original Citation	Shanley, M., Murphy, J. and Molloy, P. (2012) 'Offshore wind farm service vessel, hull design optimisation', 4th International Conference on Ocean Energy, Dublin, 17-19 October.
Type of publication	Conference item
Link to publisher's version	https://www.icoe-conference.com/about-icoe/conferences/icoe-2012-dublin/
Rights	© 2012 The Authors.
Download date	2026-10-07 10:21:06
Item downloaded from	https://hdl.handle.net/10468/7342

UCC
University College Cork, Ireland
Coláiste na hOllscoile Corcaigh

Offshore Wind Farm Service Vessel, Hull Design Optimisation

M. Shanley[1], J. Murphy[1], and P. Molloy[2]

[1]Hydraulics and Maritime Research Centre, Civil and Environmental Engineering
University College Cork, Youngline Industrial Estate, Pouladuff Road, Togher, Ireland
Email: m.shanley@student.ucc.ie & jimmy.murphy@ucc.ie

[2]Mechanical and Biomedical Engineering, College of Engineering and Informatics,
National University of Ireland Galway, Ireland
Email: padraig.molloy@nuigalway.ie

Abstract

Access to a wind turbine is a major issue, currently there is a 1.5m significant wave height (Hs) limit for the standard "step over" method for transferring personnel to an offshore wind turbine. According to the Carbon Trust being able to access, wind turbines at a wave height of 3m would be worth £3 billion to the offshore wind industry. The current research addresses this issue by examining a novel multihull design concept for an Offshore Wind Farm Service Vessel. The objective of this work is to carry out a feasibility study of the proposed design as previous research indicated that the design reduces the heave and pitch motions by dampening its response to the wave motion.

The proposed design is analysed with both hydrostatic and hydrodynamic analysis using the computational fluid dynamic (CFD) package ANSYS CFX. Also to be undertaken is physical model testing of the analysed design in the National Ocean Test Facility's wave basin to determine the accuracy of the computational analysis.

Keywords: Offshore Wind, Operation and Maintenance, Personnel Transfer, Wind Farm Service Vessel

1. Introduction

When accessing offshore wind turbines in harsh sea conditions, the vessels utilised must keep their motion minimised in order to operate safely. The accelerations induced on the vessels hinder the transfer of personnel from vessel to wind turbine as well as the operation of a crane for the transfer of replacement parts. In addition, when operating a wind farm it is extremely costly to have wind turbines broken down and unable to produce electricity By increasing the weather window that a vessel can get service personnel on and off the wind turbine, directly increases the wind farms output.

This paper proposes a novel new design. The concept is to have a hull composed of a number of buoyant tubes. The buoyant tubes damp the vessel's motions due to the viscous action of the water around the tubes. See Fig. 1 for a concept drawing of the design.

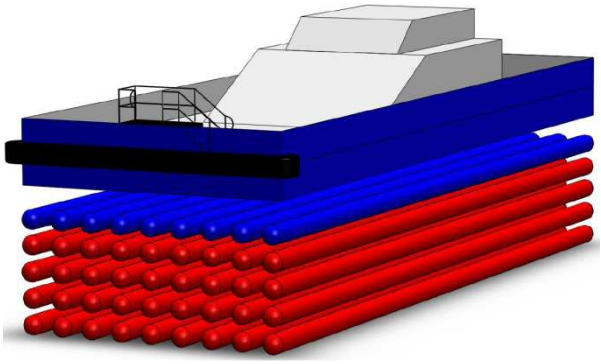

Figure 1. Concept Model Design

2. Initial Physical Testing

Initially physical testing was carried out of the model concept to investigate the possible potential of the concept design. A 1/50th scale model of an offshore supply vessel (LOA 80m, beam 20m displacing 3000tons) was tested and a monohull vessel of the same size was also tested for comparison. Fig. 2 below shows the initial testing of the model.

The model was comprised of a series of sealed tubes supported in a frame. Testing results shown here were obtained using Bretschneider wave energy spectra (T_p 8.5s, H_s 4.8m, and T_p 12.7, H_s 4.6m). The vessel's response to the wave spectrum is reduced when compared to a monohull's response.

Figure 2. Initial Testing of the Model Concept

Fig. 3 below shows the reduction in Pitch RAO from monohull to the Tubular Multihull design, particularly in the 6 – 12 second range. In the 8 second wave period region the Tubular Multihull RAO is of the order of 35% that of the supply vessel monohull design and is of the order of 20% in the 6-7 second range. The wave tank data shows that the pitching RAO reduction recorded drops back as the wave period increases but remains significant.

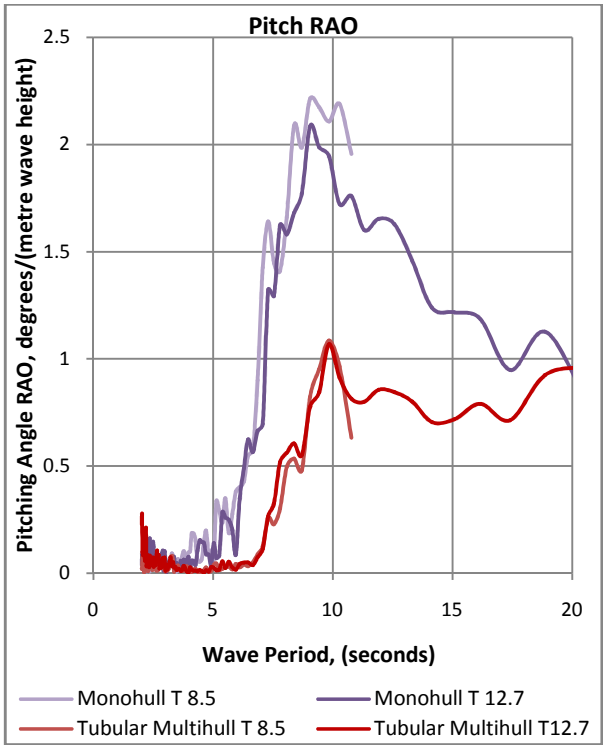

Figure 3. Pitching Angle RAOs obtained in tank testing standard monohull and Tubular Multihull models.

Fig. 4 shows the reduction in Heave RAO from monohull to the Tubular Multihull. The reduction follows a similar trend as pitching, with a higher reduction in shorter waves and the reduction dropping below 50% as wave period increases.

In heave, pitch and roll a significant advantage is observed with the Tubular Multihull design as regards the initiation of significant motion – unlike the monohull the new design shows very high stability up to a wave period of 8 seconds and thereafter remains significantly better as the wave frequency decreases.

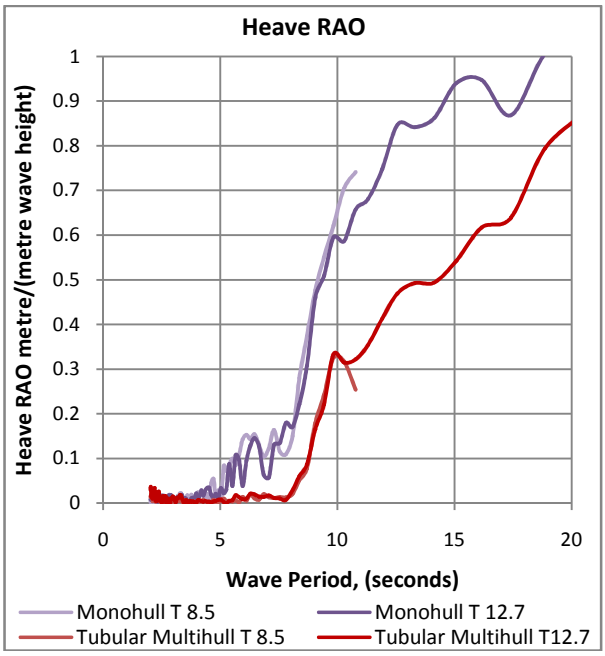

Figure 4. Heave RAO's obtained in tank testing standard monohull and Tubular Multihull models.

2.1 Conclusion of initial physical testing

The Authors concluded from the initial physical testing that the design has merit. The RAO's are significantly reduced in the 6 – 12 second range, which encompass the normal operating conditions of the North Sea.

Based on the physical testing, static stability calculations, regulations and standards a refined design was determined. This design is being modelled using the Computational Fluid Dynamics code ANSYS CFX.

3. Updated Vessel Specification

The vessel is intended to be wind farm service provider category 1 as outlined by the Det Norske Veritas, (DNV) regulations. These requirements limit the vessel to a length of 24m and the maximum number of passengers to 12.[7]

The standard wind farm, service vessel has a beam of 8m and a displacement of 65 tonnes. Hence, the concept design is being analysed with these parameters fixed.

A design that met the above requirements and was statically stable which also adhered to the fundamental design concept, resulted in a design with the following parameters as illustrated in Fig. 4:

- Horizontal spacing (S_h) 1.259 metres
- Vertical spacing (S_v) 0.9 metres

- Number of tubes vertically 4
- Number of tubes horizontally 7
- External radius of tubes 0.225 metres
- The top of the deck is 2.938 metres above the waterline.
- Draft is 1.987 metres
- Angle of Deck immersion is 23 degrees

The designed total mass of the vessel as stated earlier is to be 65 tonnes. At this early design stage, this was broken down into three components: Firstly, the buoyant tubes and associated supporting structure at 39 tonnes, secondly the deck structure at 15 tonnes and thirdly a cargo of 11 tonnes.

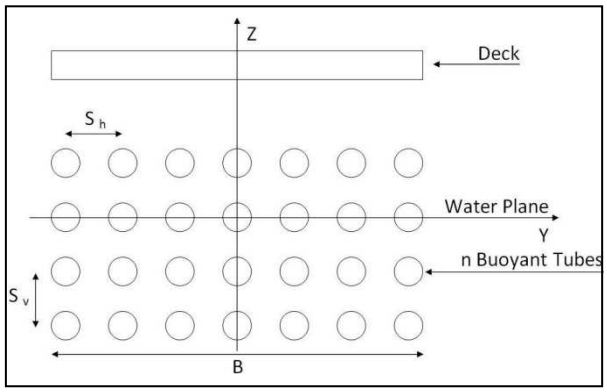

Figure 5. Beam view showing the different variables

4. Static Stability Analysis

The static stability of the craft can be analysed under the category of a multihull craft in Annex 7 of the 2000 HSC Code.[1]

In particular the area under the GZ curve should be at least 0.055 x 30°/θ. (where θ is in this case the angle of deck immersion) From the GZ curve presented in Fig. 6 the area under the curve is 0.1113 m.rad which is greater than 0.07122 m.rad (from the equation)

The HSC Code also states, "The maximum GZ value shall occur at an angle of at least 10°" which it clearly does as it is still increasing at 23°.

The metacentric height (graphed against roll angle in Fig. 7) is initially quite large, falls, and rises as the geometry of the water plane area changes with the vessel's roll angle. This effect is also present on the GZ curve, Fig. 6. The effect of this, if any, on the roll motions and roll period will be assessed during the upcoming physical testing of the vessel.

The metacentric height is considerably larger than most vessels in the North Sea and so the vessel's roll period will likely fall outside the spectral range of periods for North Sea conditions.

The longitudinal metacentric height GM_L is 52.6 m, the pitch period is affected by the GM_L and the greater this value can be the better.

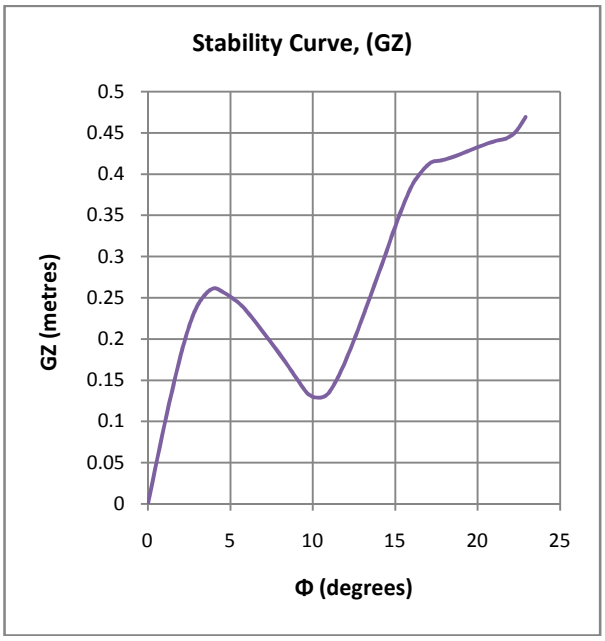

Figure 6. GZ Curve, up to the point of deck immersion

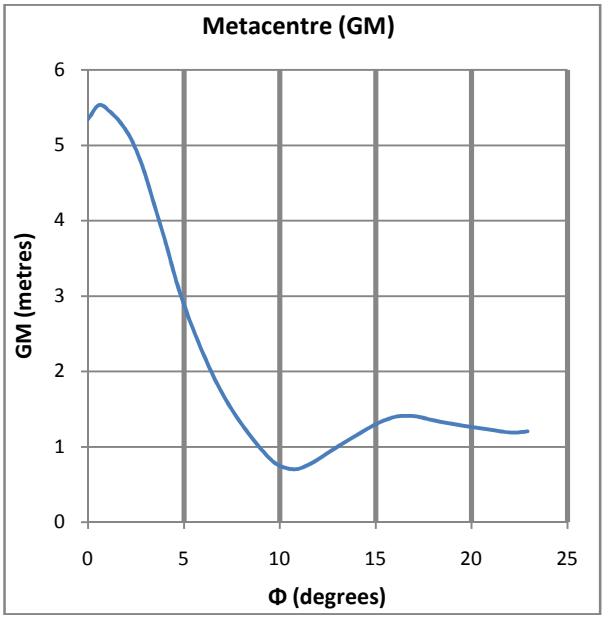

Figure 7. Metacentric Height, up to the point of deck immersion

5. Computational Fluid Dynamic Analysis

Ansys CFX was used to model the vessel because of the nature of the concept being modelled meant that the assumptions for frequency domain analysis with potential flow theory were not valid in this case.

Firstly, the water plane area of the vessel changes greatly with small angles of roll and pitch and secondly the concept sets out to use viscous effects to reduce the vessels motions. Hence, analyse numerically the concept presented, a software package that computed the vessels movements in the time domain and accounted for viscous flow was required.

Computational fluid Dynamics (CFD) met that requirement as it computes the full Navier-Stokes equations. Ansys CFX was chosen as it has incorporated algorithms that can compute a free surface and the movements of a floating body in the fluid. In addition, it is industry proven software.
CFD analysis is a time and computationally expensive exercise, but a requirement if this concept is to be adequately modelled numerically. CFX is not at a stage to replace physical model testing. It however can add to physical tests and expedite the process of optimising the design

The fluid modelling software Ansys CFX solves the unsteady three-dimensional Reynolds Averaged Navier-Stokes Equation (RANSE) for simulating a 3-D numerical wave tank and floating object. The general-purpose RANSE solver Ansys CFX, which is based on the Finite Volume Method (FVM) was used for the present simulation. Multiphase simulations for free surface deformation were computed using Volume of Fraction (VOF) method. The movement of the vessel was computed using the rigid body solver incorporated in Ansys CFX.

5.1 Model Setup

Fig. 7 shows the domain setup. There is a flap type wavemaker on the left that generates waves according to the following formula.[2]

$$\frac{H}{S_0} = \frac{4\ \sinh kh}{\sinh 2kh + 2kh}\left[\sinh kh + \frac{1-\cosh kh}{kh}\right]$$

There is an opening boundary at the top, which allows air to enter and exit as required as the waves oscillate. There is a parabolic at the end of the wave tank to dissipate the wave energy by means of wave breaking. A parabolic beach was found to be most effective at this, whilst keeping the domain size to a minimum. This is a full scale simulation with a domain of 500m long, 75m high and a water depth of 50m. The model allows 150m for the waves to fully form and allow for the initial exponential decay. There is then a 50m section to place the model in. The model is shown in Fig. 7 inside a circle. The circle and other lines shown inside the domain are fluid to fluid boundaries to aid mesh optimisation.[3,4]

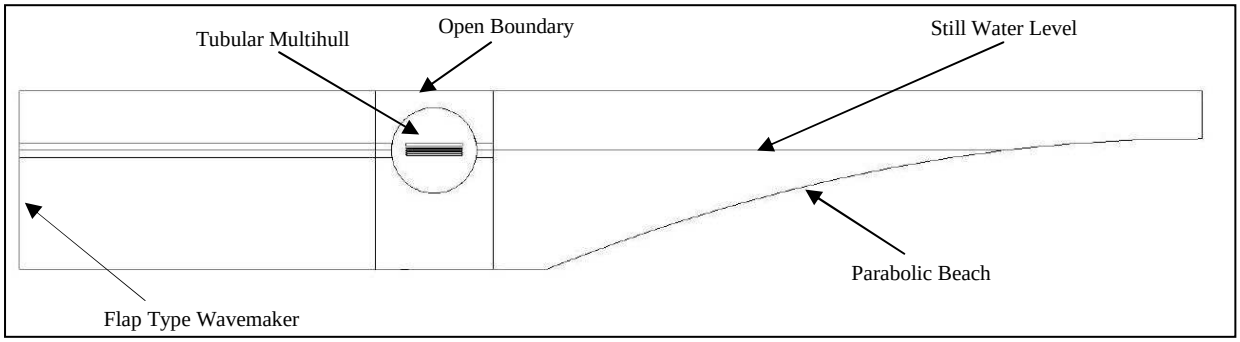

Figure 8. Domain Setup

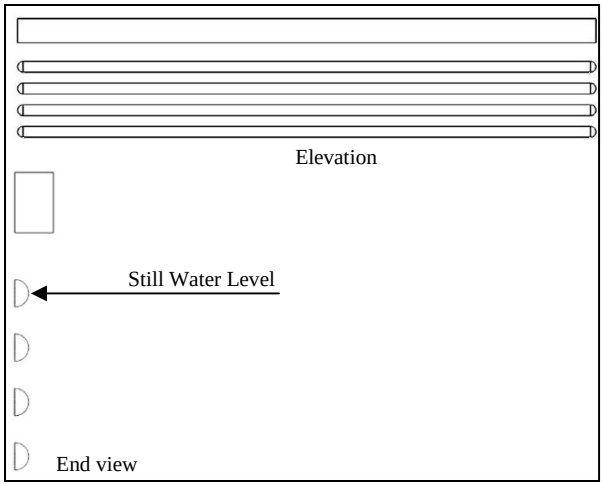

Figure 9. Elevation and End view of the Rigid Body

Symmetry was utilised in the model to keep the mesh size to a minimum for a three dimensional simulation. Fig. 9 opposite shows an elevation and end view of the section of the rigid body that was tested. (Ansys CFX refers to any floating object as a rigid body and computes it's movements with a rigid body solver.) The thickness of the entire domain is 1/14 of the vessel width. This results in a half cylinder and half the spacing between cylinders, with an overall domain thickness of 0.5 x S_h = 629.5mm

The front and rear faces of the simulation have a symmetry boundary condition, also the vessels movements are restrained to 3 degrees of freedom, Heave, Pitch and Surge.

5.2 Model Mesh and Timestep setup

The simulation has 2,028,443 elements, Fig. 10 right and Fig. 11 below, show how the arrangement of the mesh elements. The mesh is refined at the water surface, to prevent what is known as "numerical damping" where the wave height diminishes as it propagates. This numerical damping is effected by the timestep also and it is recommended that the timestep be 1/100 of the wave period[5]. In the simulation presented, a timestep of 0.05s was used for a 6.1-second wave period.

A very small timestep coupled with a very large number of mesh elements results in a very long simulation time. For this reason CFX models are time-consuming operations, however the output is very detailed. On an Intel Xeon, 16 processor @ 2.27 GHz with 96 GB of RAM on a 64-bit operating system a computation time of 1.7 hours per second of simulation was required for the presented simulation.

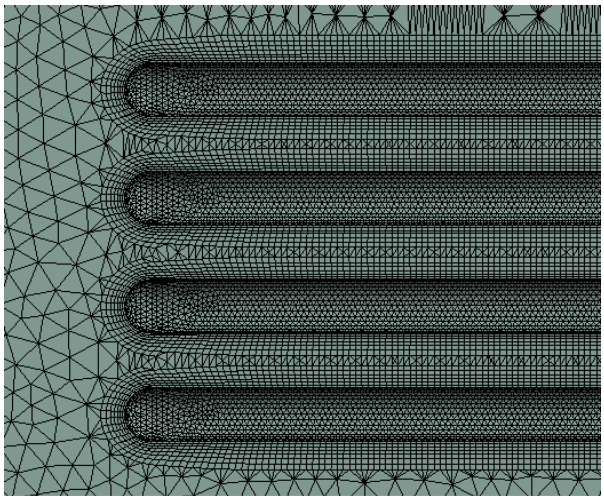

Figure 10. Detailed view of mesh around tubes

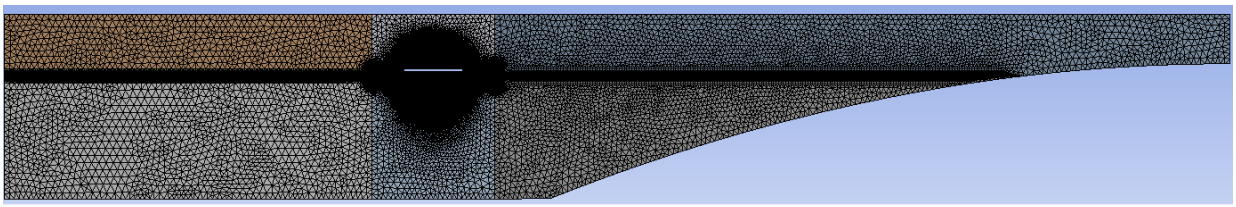

Figure 11. Mesh Density of the CFX Domain

5.3 Simulation Results

The results presented in this paper consist of a calibration model and the output from the current simulation.

To bring the model to this stage, mesh and timestep sensitivity analysis was carried out both on the calibration model and on the model containing the rigid body. Due to the nature of the flow around the tubes and the forces induced on them, the coupling between the rigid body solver and the fluid solution was enhanced to achieve a convergent solution. Relaxation of the mesh motion was also required.

The presented results are for a simulation of a 3 metre wave with a 6.1 second period.

To determine the accuracy of wave propagation throughout the tank, a simulation was run without rigid body in the domain. The results of this are presented in Fig. 12. It is also demonstrated in the same graph that the wave height in the simulation, which includes the rigid body, is higher. This is likely due to radiated waves from the rigid body itself. To determine the extent of radiated waves a longer simulation is required and the rigid body should be placed further from the wavemaker to take sufficient readings to measure this.

The effects of this wave on the rigid body are shown in Fig. 13 showing the heave motion and Fig. 14 showing the pitching motion. Relative to the size of the wave, the induced motions are larger than that demonstrated by the original model, due mainly to the relative size of the vessel to the wave.

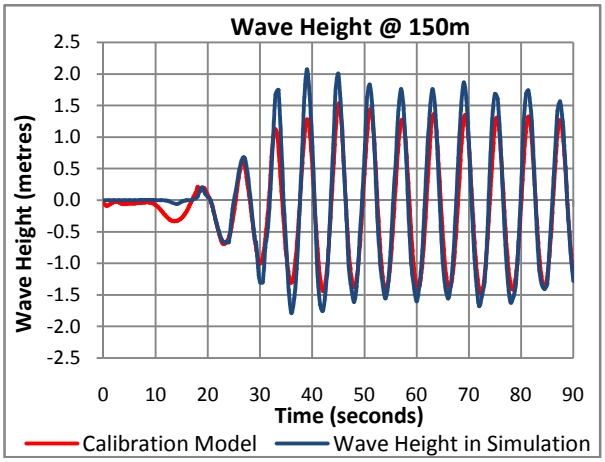

Figure 12. Wave Height

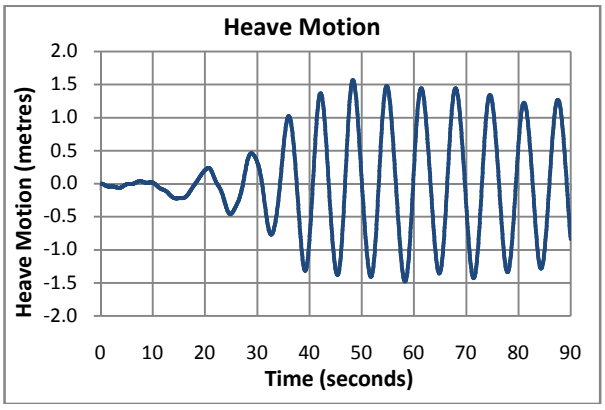

Figure 13. Time Series of Heave Motion

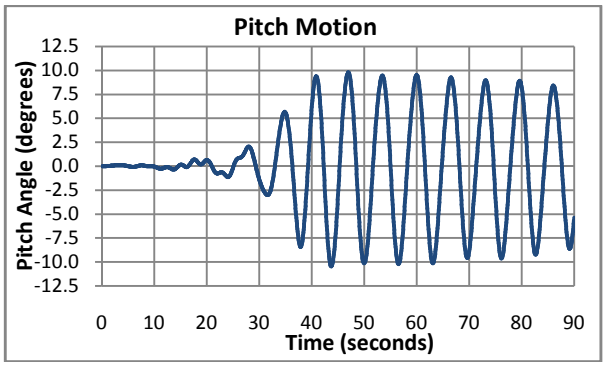

Figure 14. Time Series of Pitch Motion

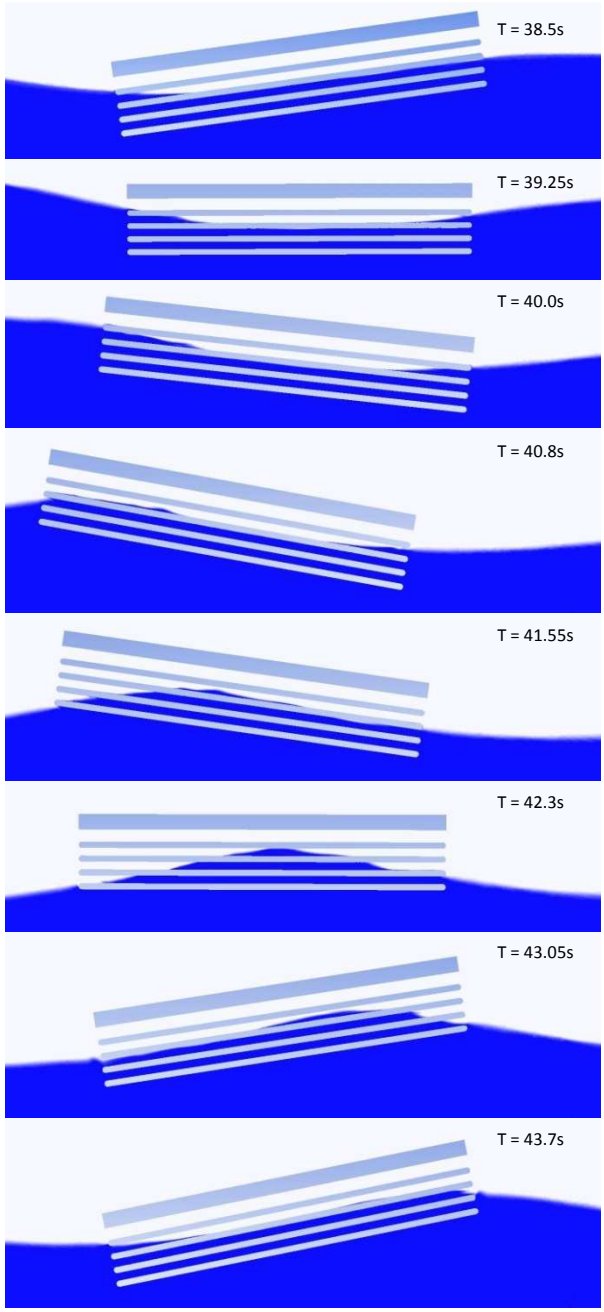

Figure 15. Vessel motion throughout a wave

Fig. 15 previous shows a series of images that track the vessels movements as it encounters a wave with an incident wave height of 3.67m and a period of 6.1 seconds. The vessel is following the wave profile in this situation. This motion is unsatisfactory as the pitch and heave motion is quite large relative to the wave, these can be seen clearly in Fig. 13 and Fig. 14. Further testing will determine if a change the configuration of the design will be required to reduce the vessels response amplitude operators.

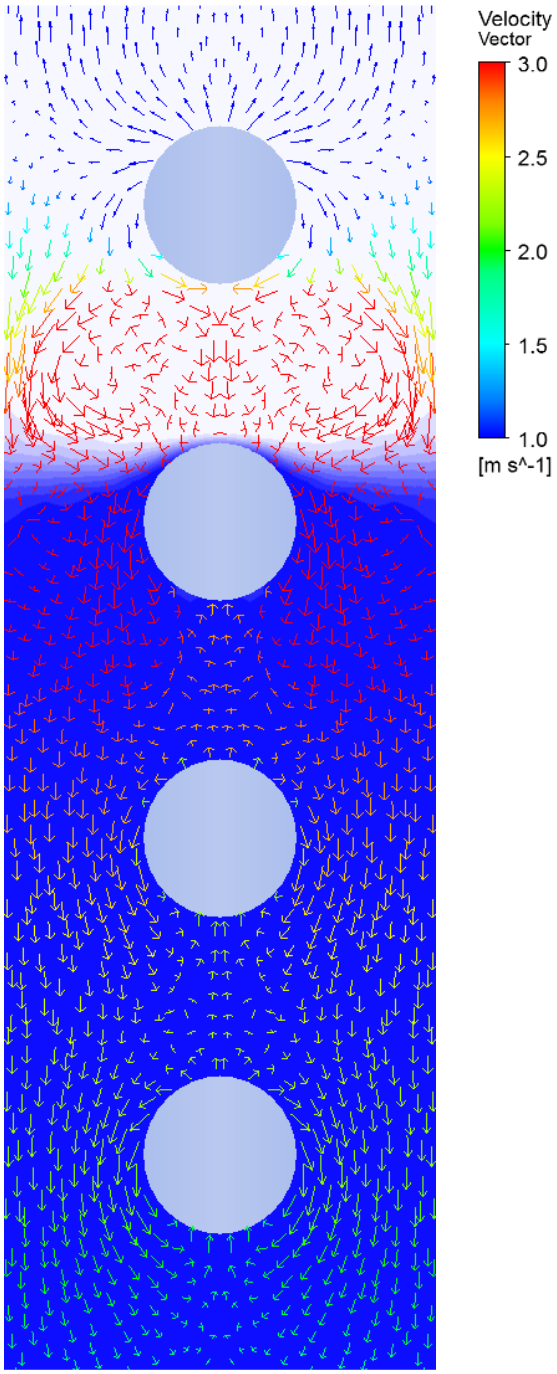

Figure 16. A Section at 4.251m in the –x direction from the Centre of Gravity showing the velocities of the water around the tubes at 41.55 seconds

Fig. 16 on the previous page is a sample section through the model at a location of 4.251m in the negative x direction from the Centre of Gravity of the vessel. The image shown has been mirrored once to give a clearer picture of the flow around the tubes.
The velocity is quite high reaching 3 $m.s^{-1}$ and naturally decreases with depth. There are small areas above and below the tubes with velocity normal to the tubes, they represent an eddy moving with the same velocity as the tube at that point. The velocity of the air is quite high near the free surface, as is expected as the water is moving with a similar velocity at that location, but air is compressible and hence reduces rapidly, also it has a negligible effect on the vessels motions as along with being compressible also has low density.

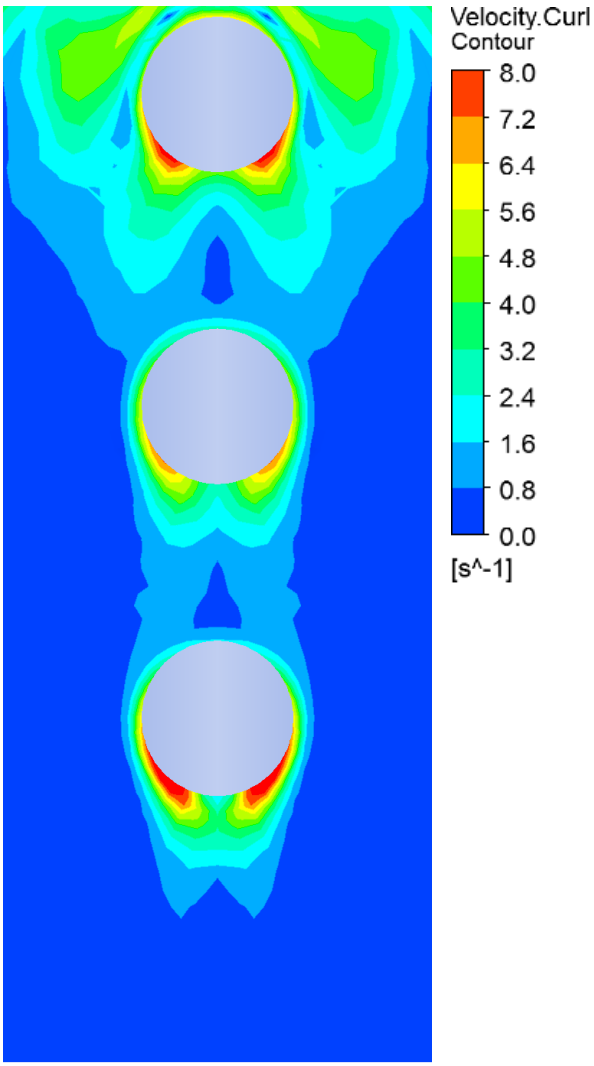

Figure 17. An image showing the curl of the velocity or vorticity of the water around the tubes at 41.55 seconds on a section 4.251m in the negative x direction from the Centre of Gravity

Fig. 17 above shows the instantaneous water vorticity at 41.55 seconds at a location of 4.251m in the negative x direction from the Centre of Gravity of the vessel. Vorticity is a measure of the amount of rotation in a fluid and is computed as the curl of the velocity field and hence referred to as the velocity Curl in Fig.17. The graph of vorticity enables the visualisation of the vortex shedding pattern. In order to maximise the heave and pitch damping then vortex shedding must be optimised.[6]

6. Summary

The physical model testing showed a significant improvement over a conventional vessel of the same size. There will be a market for an improved offshore wind maintenance vessel, due to the increase in maintenance required for the UK's upcoming round three projects. Therefore, a design for such a vessel was formulated based on the existing codes and static stability calculations.
The concept in this situation with this arrangement of tubes does not appear to have a noticeable effect compared to that of a conventional vessel. These preliminary results suggest that a change to the configuration of the vessel's design is required, however further simulations need to be run to confirm this. The difference in size, most notably length, between the physical model and the CFD simulation has negatively affected the vessels motions.
The work to date has allowed the HMRC to develop expertise in the field of CFD modelling, and to gain an implicit understanding of what the software's advantages and disadvantages are. The CFD analysis has proved to be a time and computationally expensive exercise.
More generally, the following conclusions can be made from the work carried out to date.
CFD is particularly useful where the water plane area is constantly changing, as in these situations time domain analysis is required and to some extent, the viscous effects may be too. Frequency domain analysis of potential flow is not possible in these situations as the assumptions for this analysis are invalid.
CFD and specifically Ansys CFX is not at a stage to replace physical model testing of this type of vessel as the simulation needs to be validated with a physical model. Careful consideration should be taken before attempting CFD analysis as it is an expensive and time consuming process. Therefore, CFD should only be used when other software cannot effectively carry out the required analysis.

7. Further Work

The work carried out to date has indicated that continuation with the CFD analysis is worthwhile, that physical testing will be necessary. It is intended to continue the CFD analysis in regular waves to determine the RAO's for a number of different wave conditions. Following that, a number of irregular wave conditions will be investigated.

It is intended to test a 1:25 scale model of the vessel in the HMRC's wave basin, to determine if the CFD model accurately approximates the physical model.

The model will also investigate the natural periods of the vessel in Heave, Pitch, and Roll.

In addition, it is intended to place it behind a scale model of a wind turbine to determine the effect that will have on the vessels motions.

Acknowledgements

The Authors wish to acknowledge the Graduate Research Education Program in Engineering and the HMRC for the financial support of the project.

References

[1] The Maritime and Coastguard Agency, (2009). International code of safety for high-speed craft, (2000), 200HSC Code, 2009.

[2] R.G. Dean, and R. A. Dalrymple, (1984). Water wave mechanics for engineers and scientists, Prentice-Hall Inc., New Jersy.

[3] W. Finnegan, J. Goggins, (2012). Numerical simulation of linear water waves and wave-structure interaction, Ocean Engineering.

[4] A. Lal, M. Elangovan, (2008). CFD simulation and validation of flap type wave-maker, World Academy of Science, Engineering and Technology.

[5] M.C. Silva, M. de Araujo Vitola, W.T. Pinto, C.A. Levi, (2010). Numerical simulation of monochromatic wave generated in laboratory: Validation of a cfd code, 23º Congresso Nacional de Transporte Aquavia¡rio, Construcao Naval e Offshore, Rio de Janeiro.

[6] L. Tao, B. Molin, Y.M. Scolan, K. Thiagarajan, (2007). Spacing effects on hydrodynamics of heave plates on offshore structures, Journal of Fluids and Structures.

[7] D.N.V. Det Norske Veritas, (2012). Tentative rules for service craft carrying up to 12 passengers.