SIMULATING WAVE ACTION IN THE WELL DECK OF LANDING PLATFORM DOCK SHIPS USING COMPUTATIONAL FLUID DYNAMICS

Don Bass, Memorial University of Newfoundland, Canada
David Molyneux, National Research Council, Canada &
Kevin McTaggart, Defence R&D Canada – Atlantic, Canada

SUMMARY

Some interesting hydrodynamics arise when a Landing Platform Dock ship with a partially flooded well deck is subjected to waves. The motion of the ship induces sloshing in the contained fluid, which in turn generates forces that affect the motion of the ship. If the fluid is only partially contained then energy can also be transferred from the external waves to the fluid inside the ship. This paper describes the development of the software tools required to simulate wave action in a flooded well deck, using the integration of a commercial CFD code with a ship motions code. Three levels of integration were considered. The most complex integration involved the full coupling of the CFD code with the ship motions code, so that ship motions, fluid motions and fluid forces are calculated at each time step. The problem can be simplified if it is assumed that the waves are small, and do not induce motion of the ship. Finally the problem can be further simplified if it is assumed that the waves are generated at the entrance to the well deck, and only internal flow is considered. Each simplification reduces the computational time significantly, but is only valid if there is a small effect on the results. This paper shows that for head waves, in wave heights and periods typical of operating conditions for LPDs, there were no significant errors introduced by the simplifications, when compared to available model data. The paper then explores how wave height within the well deck is affected by well deck geometry.

1. INTRODUCTION

Landing Platform Docks (LPDs) are ships designed to be able to transfer large numbers of troops and their equipment to shore, in areas where it is assumed that the shore-based infrastructure is non-existent. The most effective method found so far is to use landing craft as ferries between the ship and the shore. In order to make this operation as efficient as possible, a floodable well deck is built into the stern of the ship. The inboard end of the dock is typically a sloping ramp, suitable for driving vehicles from the ship into the landing craft, and the outboard end of the dock is open to the sea.

The general concept and limits of operation for an LPD are described in detail by other authors [1, 2, 3] and summarized here. While the ship is in transit, the stern gate is closed, the well deck is dry and the landing craft rest on the floor of the dock, which is above the outside water level. Figure 1 shows an LPD in transit, with the dock gate closed. When the LPD reaches its destination, ballast tanks are flooded until the water level in the dock is sufficient to float the landing craft. The stern gate is then lowered and held down by hydraulic rams. With the dock gate sloping downwards, the dock is ready for embarking and disembarking landing craft, which can occur with a stationary ship, or the ship moving slowly forwards. Figure 2 shows a landing craft within the well deck and gives a good indication of the limited amount of space in the flooded well deck. A summary of dimensions for LPDs and similar ships constructed over the last forty years is given in Table 1.

Wave action inside the well deck results from the combined effects of ship motions causing the sloshing of a partially confined volume of fluid inside the ship and external waves propagating into the enclosed space. Landing craft can operate safely in wave heights up to Sea State 4 [3]. However, even in this operational zone bore waves can occur, which are much higher than the average internal waves. Wave heights inside the well deck are generally smallest for head waves [1].

Figure 1, HNLMS Rotterdam, in transit with stern gate closed

		Ship Dimensions				Well dock dimensions			
Year	Class	Length	Beam	Draft (transit)	Displacement (transit)	Length	Breadth	Area	Country
		m	m	m	tonnes	m	m	m^2	
1965	Fearless	158.5	24.4	6.2	11,060	56.0	14.6	818	UK
1965	Austin	173.8	30.5	7.0	16,500	120.1	15.2	1826	USA
1965	Ouragan	149.0	23.0	5.4	8500	120.0	13.2	1584	FRA
1969	Anchorage	168.6	25.6	6.0	13,700	131.1	15.2	1993	USA
1985	Whidbey Island	185.8	25.6	6.3	15,726	134.1	15.2	2038	USA
1987	San Giorgio	137.0	20.5	5.3	7,665	20.5	7.0	144	ITA
1988	Bougainville	113.5	17.0	4.3	4,876	78.0	10.2	796	FRA
1990	Foudre	168.0	23.5	5.2	12,400	122.0	14.3	1745	FRA
1998	Rotterdam	166.0	25.0	5.9	12,750	56.0	14.6	818	NL
1998	Osumi	178.0	25.8	6.0	8,900	55.0	14.0	770	JPN
1998	Galicia	160.0	25.0	5.9	13,815			885	SPA
2001	San Antonio	208.4	31.9	7.0	25,300	55.0	14.0	770	USA
2002	Albion	176.0	28.9	6.6	16,981	60.0	15.0	900	UK

Table 1, Summary of overall dimensions for LPD type ships

Since wave action inside the well deck limits the operation of the landing craft, it is desirable to minimize the internal wave height over the widest range of conditions. The loading ramp at the inboard end of the dock can be used to provide some wave absorption, in much the same way as a beach. Also alternative structural arrangements for the well deck can provide different degrees of damping. For example, wing tanks, porous screens and damping tanks have been tried to minimize the internal wave height.

Figure 2, Single landing craft inside well deck of HNLMS Rotterdam

Most publications on LPD hydrodynamics have described extensions to ship seakeeping research, using models of approximately 1:40 scale. Results from a typical model test program that was carried out for investigating wave action inside the well deck of an LPD are given in [1]. Models of two ships were tested. The first one was a ship in service considered to have good operational characteristics and the second was the proposed new design. The models were tested in two irregular wave spectra, with different modal periods, over a range of wave headings and speeds. The experiments were used to assess the effectiveness of different dock arrangements on the new ship in two ways. One assessment was based on minimizing the measured wave height within the dock (with no landing craft present) and analysis focused on ship heading and speed as well as different structural arrangements for the well deck. The second assessment focused on the ability of a coxswain to dock a radio-controlled model of the landing craft without a collision with the ship. Since human reaction is difficult to scale, the basis for comparison was that the new design should be no worse at model scale than the model of the ship in service. Other researchers have used variations on this technique [2, 4] and it represents a practical method of modelling the complete system.

Model experiments are a relatively expensive option for predicting wave action within the well deck. Tools are required that will enable the effective comparison of wave height within different dock structures, especially in the early stages of design development, without carrying out model experiments. This paper discusses the requirements for simulating wave action in a flooded well deck, using the integration of a commercial CFD code with a ship motions code. It then describes the validation of the different numerical approaches against available data from model experiments. Finally the paper presents comparisons of predicted wave amplitudes within several different well deck designs. The results described in this paper were the result of a collaborative research project between Defence R&D Canada – Atlantic, the United States Coast Guard, the Royal Netherlands Navy and the National Research Council of Canada's Institute for Ocean Technology.

2. DEVELOPMENT OF THE NUMERICAL APPROACH

2.1 COMPONENT COMPUTER CODES

Simulating the sloshing of a fluid inside a closed tank can be carried out using commercial CFD software. The code chosen for this research was Flow-3D, developed by Flow Science Inc. This is a general purpose CFD program, initially developed for modeling the sloshing of fluids within liquid fuel tanks for rockets. One of the attractive features of Flow-3D was its ability to simulate non-linear, transient flows in the time domain. Within Flow-3D, the Navier-Stokes equations are discretized using the Volume of Fluid approach. A full Navier-Stokes solver was used since we considered it important to be able to model the energy losses associated with viscous flow around porous ramps, baffles and other similar structures. Without this requirement it might have been acceptable to simulate the fluid within the well deck using simpler solvers.

Within the CFD program, the boundary between two fluids (air and water in the case of the LPD) was calculated using a surface capturing technique, which allowed for very large distortions of the fluid from the horizontal. This enabled the simulation of steep and breaking waves. A range of boundary conditions could be specified for structure and fluid boundaries within the model, including walls and specified pressure and velocity within the fluid. Another useful feature was the use of a non-inertial reference frame. This technique was efficient for simulating sloshing, since it allowed the direction of gravity forces to vary, within the fixed geometric grid used to describe the structure.

The structure was modelled using a fixed (Eulerian) grid of rectangular control elements. The fixed rectangular grid system allowed for rapid set-up of numerical models of complex geometries. Such grids are relatively computationally inefficient, but for a typical well deck, the geometry was sufficiently simple for the inefficiency not to be significant. Another limitation was that the definition of the geometry was also limited, but again this was not important for the simple geometric structures of a typical well deck.

The ship motions were predicted with MOTSIM, a three-dimensional, non-linear time domain program [5]. There were some technical issues to overcome in coupling the codes. The major difficulty was that both codes performed the integration of the motion equations to forward step the motions of the rigid body. This created difficulties since both codes adjusted the time steps within the solution process to solve the equations of motion. Clearly, the time base for each code had to be synchronized. Since the source code for Flow-3D was not available, MOTSIM had to be made a subroutine of Flow-3D. This meant that MOTSIM had to be called at each time step for Flow-3D. Also, the algorithm for tracking the orientation of the rigid body within Flow-3D suffered from drift problems in the rotational matrix. MOTSIM makes use of a natural interpolation routine associated with the linear multi-step differential equation solver to supply Flow-3D with the motion data needed on demand.

The results presented in this paper focus on waves inside the well deck of an LPD. The application of the coupled codes in this situation was the extension of earlier work done using the same two codes. The earliest use of the two codes coupled together was to model the behavior of fishing vessels fitted with fluid filled anti-rolling tanks. The same approach was used to model the behavior of water trapped on the deck of a fishing vessel [6]. Fluid behavior in moon pools and water flooding on and off the deck of intact and damaged ships had also been investigated using the same two codes for proprietary projects. The extension of this previous work to the case of fluid flowing within a ship's structure, which at the same time was open to the sea, was a strong motivation for undertaking this research.

Validation of the simulations against physical measurements was important to the success of the project. The Royal Netherlands Navy provided detailed data from model experiments that could be used for validation. The experiments were carried out as part of the development of an LPD for the Royal Netherlands Navy and a summary of the experiments is given in [1]. These experiments included measurements of wave height within the well deck for irregular waves, with a significant height of 2m and modal periods of 5 and 9 seconds. The speed range covered zero to 9 knots, for head and bow quartering waves. Structural variations of the well deck model included different gate arrangements and the use of wing tanks. Photographs of the model are given in Figures 3 and 4. The hull geometry and well deck geometry were taken from [1]. The model data was collected as part of the development of HNLMS Rotterdam. The summary dimensions for 'Rotterdam' are given in Table 1.

2.2 COUPLED MOTION OF SHIP AND FLUID

The most realistic approach to numerical simulation was to include the coupling effect between the motion of the ship and the resulting wave field inside the well deck. In this case, the numerical model treated the flow as two domains. The external domain covered the waves outside the ship and the internal domain covered the water within the well deck. The two domains were matched at the boundary to the well deck based on relative displacement (vertical and horizontal) and fluid velocity across the boundary.

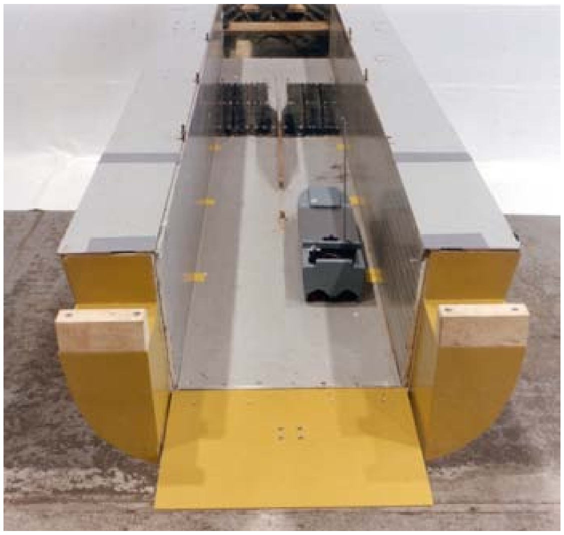

Figure 3, Model of ship with well deck, including landing craft

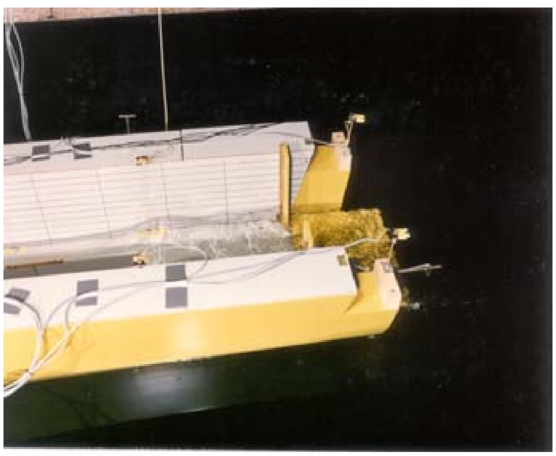

Figure 4, Model of ship with flooded well deck

In order to predict the relative motion at the entrance to the well deck, an additional refinement was required because the ship motion code did not include adequate modelling of the 'sheltering' effect caused by the waves being diffracted by the ship's hull. The CFD code was used to predict the attenuation of the waves assuming that the hull was stationary. Relative motion at the dock entrance of the moving ship was predicted based on the motions calculated by MOTSIM. For the CFD model of the waves inside the well deck, an empirical correction to wave amplitude for the 'sheltering' effect of the hull was included. No changes were made to the phase of the wave at the dock entrance in relation to the ship motions.

The fully coupled condition considered a floating ship moving in waves. The wave action for the water inside the well deck was coupled with the external wave field and the motion of the ship. The motion of the water inside the well deck included effects of the waves propagating from the external flow and the acceleration components due to the motion of the ship. The resulting sloshing forces were fed back into the ship motion program, and the responses at the next time step were predicted.

The first level of simplification was to decouple the motion of the fluid within the well deck from the forces acting on the ship. In this case, it was assumed that the motion of the ship and external waves affected the fluid inside the well deck, but that the forces due to sloshing have no effect on the ship motions. All other conditions were the same as for the fully coupled condition. This approach gave an indication of the magnitude of the sloshing forces and their effect on the ship's motions, when compared to the fully coupled simulations described above.

2.3 STATIONARY SHIP, HEAD WAVES (HEAD SEA STATIC)

In most cases, the ship is operated with its bow into the waves, and so the external waves do not flow directly into the dock. The relative motion between the ship and the water surface at the dock entrance causes waves inside the well deck. The process is complicated by the ship structure ahead of the entrance to the dock, which will tend to shelter the dock entrance from the external waves. Also the ship's structure will diffract the waves. The process will be dependent on wave frequency, with low amplitude, high frequency waves more affected by the proximity of the ship than low frequency, high amplitude waves.

A simplified version of this situation can be simulated using the CFD code alone, by assuming that the waves inside the well deck were created by the motion of the fluid at the entrance to the well deck and were not affected by the movement of the ship. In this case the hull was fixed at its normal waterline. Regular head waves were numerically generated ahead of the ship. The waves flowed along the ship's hull and the wave pressure at the entrance to the well deck created waves, which propagated along the well deck towards the bow. The waves were simulated for several wavelengths downstream from the hull. The computational domain for this approach was large, since it included the external and internal domains in the CFD code. As a result, it was time consuming to run.

2.4 WELL DECK ONLY (STATIC)

The simplest case of all was to assume that the well deck can be separated from the ship, and only the internal flow in the well deck was considered. The advantage of this approach was that it required the smallest computational domain within Flow-3D. In this case, waves were generated at the entrance to the well deck. As a result they were effectively following waves for the ship. A correction for wave amplitude, between the external head waves and the internal following waves that varied with wave frequency was included. This option would be useful for comparing different well deck designs within one ship, provided that the external flow patterns were the same, since the external flow only had to be computed once. An example of the output from the CFD code is given in Figure 5, which shows water elevation along the centerline of the well deck, together with velocity vectors. Wave elevations were computed at fixed locations within the well deck. A typical time history of wave elevation at a location 40m inside the well deck is shown in Figure 6.

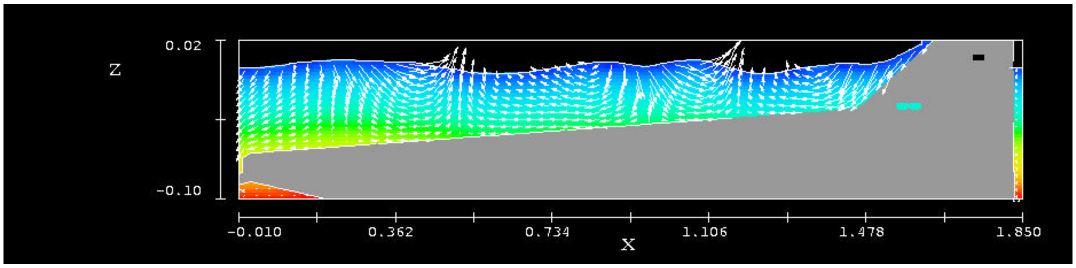

Figure 5, Wave action in the well deck modeled with Flow-3D using 'static' approach with following waves at entrance to well deck.

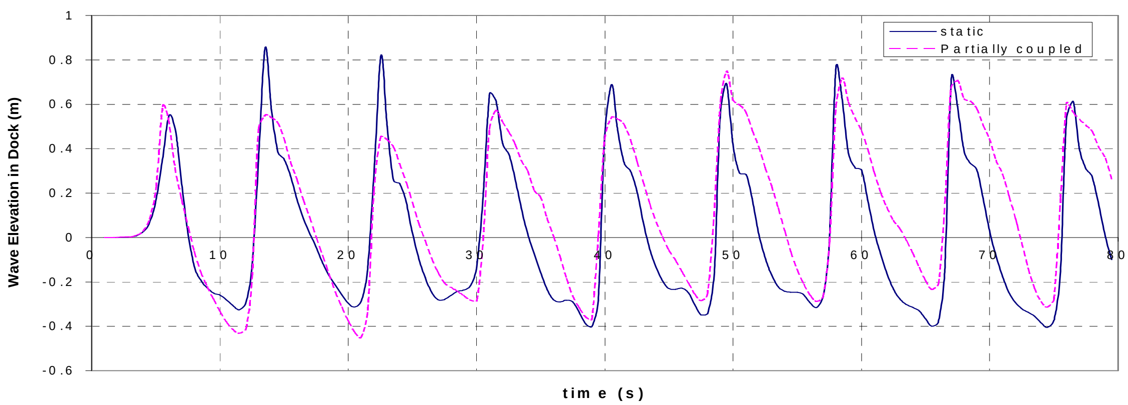

Figure 6, Time history of wave elevation within well deck, wave period =9 seconds, location=40m inside entrance of dock.

3. DISCUSSION OF RESULTS

3.1 RELATIVE MOTION AT DOCK ENTRANCE

Relative motion between the ship and the water surface at the dock entrance is an important factor determining the wave amplitude within the well deck. Relative motion at the transom was calculated for the ship at zero speed using MOTSIM. Here, the waterline for the ship was based on a flooded well deck, but the mass within the well deck was treated as a solid lump. Within MOTSIM, no account was taken of the distortion of the waves due to the presence of the ship, although forces on the hull due to diffraction effects were included. The resulting Response Amplitude Operator (RAO) is shown in Figure 7.

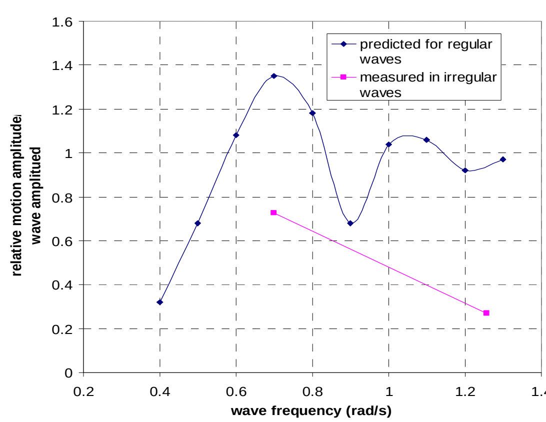

Figure 7, RAO for relative motion at stern of LPD, for head waves and zero speed

The predicted relative motion RAO, which does not include wave distortion, shows a value tending to zero at low frequencies, where the ship will tend to move with the waves, and a value tending to 1.0 at high frequencies, where the ship will be moving very little, and the relative motion is due to the wave amplitude alone. In the central portion of the curve, the RAO varies between 1.35 and 0.68. This would indicate that the waves entering the well deck could be up to 35 percent higher than the external waves. In practice, we can expect some sheltering effect from the ship at frequencies above some threshold and these values will be reduced.

There is no experimental data from regular waves with which to validate these predictions, but the experiments in irregular waves give some insight into the results. The model data in irregular waves were used to derive an indication of relative motion at the dock entrance. Measurements were made of relative motion amplitude approximately 2 metres outside the well deck and the wave amplitude, far away from the ship. The ratio of the standard deviation of each measurement is a measure of the RAO. Values of this ratio for head waves at zero speed with a significant height of 2 metres are also shown in Figure 7, for modal periods of 5 seconds and 9 seconds. Clearly these responses are much lower than the values predicted for regular waves.

The effect of forward speed on relative motion, based on data from the model experiments is given in Figure 8. This data shows that the trend is for a reduction in relative motion with increased speed for waves with a modal period of 9 seconds, whereas for the waves with a 5 second period, the values remain almost constant. Based on the results of the model experiments, there appears to be a significant sheltering effect on the waves due to the proximity of the hull. At zero speed, the effect is greater for a modal period of 5 seconds rather than for the modal period of 9 seconds. At nine knots, there does not seem to be a significant effect of wave period.

The results of the Dutch model experiments showed that the waves generated in the well deck were highest at zero speed. The zero speed case is a realistic operating condition, and the results of model experiments showed that the motions of the ship in waves with a significant height of 2 metres were relatively small. Therefore for head waves, the assumptions required for the use of the CFD code alone (without the ship motions code) were realistic and the most conservative. This was fortunate since no practical method of including forward speed of the ship within the simulation method was found.

The reason for the reduction in relative motion RAO relative to the predictions in regular waves was not fully verified. There are several factors that might have contributed to this difference. One possibility is that there is a phase shift in the waves between the cases with the dock open and the dock closed, due to diffraction at the dock gate, but that this has little effect on the wave amplitudes created within the well deck. Another possibility is that the position at which the measurements were taken was influencing the results. The measurements were made outside the well deck, but along side the gate. It may be that waves coming out of the well deck were creating a consistent trough at the entrance to the dock. This has been observed in simulations for other situations, but the experiment data was not sufficiently detailed to verify this.

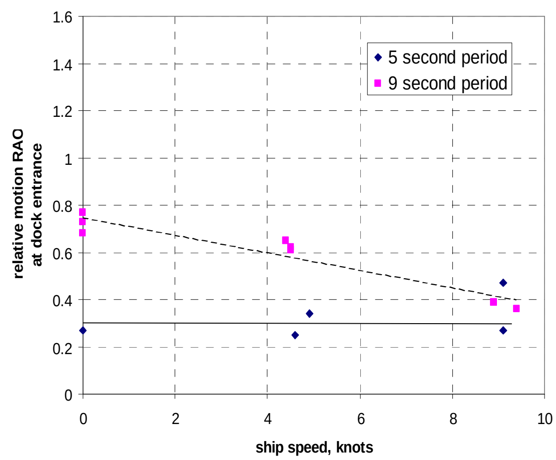

Figure 8, Effective RAO for relative motion at stern of LPD, based on measurements in irregular head waves with significant waveheight of 2 metres

3.2 WAVE HEIGHT WITHIN WELL DECK

Since the ship motions code cannot predict the wave elevations close to the hull, the 'head sea static' cases were used to simulate diffraction effects. Flow-3D was used to calculate wave heights at the dock entrance for 2 metre high waves (peak to trough) with periods of 5 and 9 seconds. In the case of the 9-second waves, there was little attenuation of the wave amplitude, and so the 2 metre waves were used as the boundary condition for the water in the well deck. For the 5-second wave case, a good deal of its energy is lost before entering the dock. A reduced wave height at the dock entrance of 0.42 metres was therefore used in the simulations where the wave was generated at the dock entrance.

Results of the simulations for head waves are given in Figure 9 for 9-second waves and Figure 10 for 5-second waves. Each figure shows the standard deviations of relative motion in the dock taken at various locations along the length of the dock for the different degrees of coupling between the two codes. Also shown are the results of experiments in irregular waves, with a significant height of 2 metres and peak periods of 9 and 5 seconds. The simulations and the model results are for zero speed.

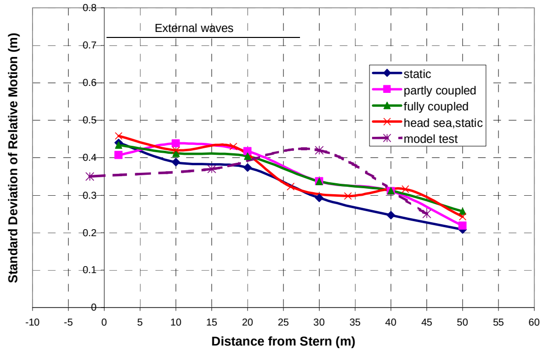

Figure 9, Standard deviation of relative motion in well deck, ship at zero speed in head waves, 9 second period

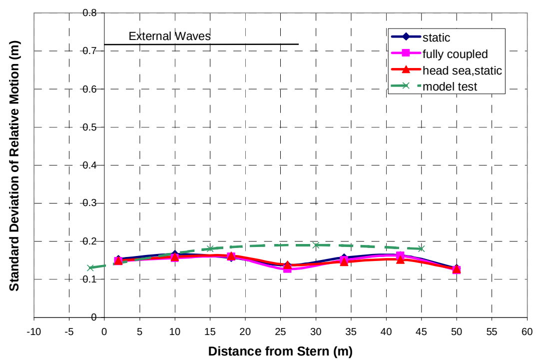

Figure 10, Standard deviation of relative motion in well deck, ship at zero speed in head waves, 5 second period

Figure 9 shows that for the particular ship being studied, there is little effect on the results between the coupled and the partly coupled case. As a result no further studies in head waves were made with the fully coupled condition. For 9-second waves, there was also little difference between the head sea static case and the static case. This indicates that the simplification of using the stern waves is reasonable for long waves. Finally the agreement between the simulations and the model experiments is good. There seems to be a shift in location of the greatest wave elevations. This could be due to differences between the reflected waves and motions of the vessel for the regular waves used for the simulation and the irregular waves used for the experiments.

Figure 10 shows a similar set of results for waves with a period of 5 seconds. As can be seen the results of the ‘static’ and ‘head seas static’ agree very well. There is also clearly even less effect of vessel motions on the waves in the dock for this short wave, as might be expected. The model results agree reasonably well except at one position (30 m). The difference in appearance of the two curves may in part be due to the lack of probes at positions where the wave elevations may be reduced (for example at 20 m).

Relative motion at the dock entrance can be discussed by considering the results at a location just inside the well deck. For the fully coupled and partially coupled cases, the results given are relative motion based on the combined movement of the ship and the water surface. For the head sea static case, the relative motion is based on the wave surface only. For the particular ship and wave conditions studied, there is little difference between the predictions of relative motion at the dock entrance and the experimentally observed values. Whilst one would expect the coupled cases to be the most realistic, for this particular ship, the combined pitch, heave and wave elevation responses show no significant difference from the case where the ship is stationary. This could be explored further by investigating different combinations of wave height and period, but there was no experiment data with which to validate the results. Note that the well deck gate was omitted from the simulations, but was present in the model experiments, which may also have affected the results.

4. WELL DECK DESIGN STUDIES USING CFD

4.1 ANALYSIS IN THREE DIMENSIONS

Based on the comparisons of the results of the different numerical models, it became apparent that for the ship dimensions and wave conditions for which validation data was available, there was little advantage to using the coupled ship motion and CFD codes. Computational times were considerably longer when the two codes were combined and the improvement in accuracy of the results was almost negligible.

The most efficient method of predicting wave heights within the well deck was to run the CFD code for the ‘head sea static’ case to obtain the diffraction effects close to the hull. The amount of attenuation of the waves close to the hull depends very strongly on the amplitude and frequency of the far field waves, with small, high frequency waves being greatly reduced in amplitude, whereas high amplitude, low frequency waves were almost unaffected by the presence of the ship. The resulting relative motion and velocity distribution can then be used as input for the ‘static’ (following seas) model of just the internal structure of the well deck. The same boundary conditions at the entrance to the dock can be used for different internal well deck structures.

One of the major issues around performing the simulations was the length of time to come to a solution. For the validation studies, a symmetrical 3-dimensional model was used on a DEC Alpha workstation with a processing speed of 600 MHz and 786 MB of memory. On this hardware, simulations were taking approximately two days to simulate approximately 600 seconds of data for each combination of wave amplitude and frequency.

This was not effective for design evaluation, at least in the preliminary stages and so attempts were made to find more practical strategies.

One factor that would vastly improve the computational time would be to reduce the degrees of freedom in the model. The 3-dimensional simulations and the experiments both indicated that most of the wave action was along the centerline of the well deck. If this was the case, then the problem could be reduced to a two-dimensional model, at least for the preliminary evaluation of design alternatives. However, for the simplifications to be justified, the 2-dimensional method must rank different dock geometries and structural features in the same order as the 3-dimensional model and the same order as physical model experiments.

The only data for validating this approach was from proprietary data outside this collaborative project. The CFD simulations of this situation showed that the 3-dimensional CFD model gave overall agreement within 20% of the physical model data, with all designs ranked in the same order between the 3-dimensional CFD simulations and physical model experiments. The 2-dimensional model ranked the different dock structure designs in the same order as the 3-dimensional CFD model, but there was only one set of model data set for an effectively two-dimensional structure.

This analysis indicated that it would be feasible to simplify the CFD simulations to two dimensions, where one combination of wave height and period could be simulated in approximately half an hour. As a result a well deck structure could be evaluated for up to sixteen wave height and period combinations in eight hours rather than several days. Using two-dimensional analysis a well deck design could be processed overnight and analyzed the following day.

4.2 DESIGN ANALYSIS IN TWO DIMENSIONS

Well deck design features were investigated using a two dimensional CFD model, with waves entering at the mouth of the well deck. This was effectively the 'static' case described above. Geometric factors considered were the effect of a sloping floor on the bottom of the well deck, the effectiveness of different beach designs and the effect of dock size on the wave amplitude within the well deck.

Each case was simulated for five wave frequencies from 5.5 to 13 seconds, and two wave amplitudes at each frequency (although not all the results are presented in this paper). The wave amplitudes corresponded to 0.39m and 0.19m full scale. These were chosen to cover the range of distortion (due to proximity of the ship) expected for a 1-metre amplitude wave in the far field. Two wave amplitudes were chosen, since it has been observed in model experiments that the resulting wave amplitudes in the well deck were non-linear with external wave amplitude.

The results of the simulations were analyzed to determine the time dependent variation of relative motion (wave elevation in the case of the static model) at fixed points from the entrance of the well deck. At each of these points, the standard deviation of the computed wave height was calculated and the location with the maximum standard deviation determined. The results were presented as response amplitude operators (RAO) defined as the maximum standard deviation of the relative motion inside the dock divided by the standard deviation of the external wave for each wave frequency. Dock designs were ranked on the basis of the magnitude of the average RAO across the whole frequency range.

4.2 a) Sloping Dock Floor

To study the effect of dock floor slope, two different 2-dimensional shapes were used. In one case the geometry was defined based on the centerline of the 3-dimensional studies, for which model test data were available. In this example, the depth of water at the dock entrance was greater than at the foot of the ramp. In the other case, the geometry was changed to represent a flat floor, with a constant water depth, ahead of the ramp of 2.07 metres. Both cases had 15-degree solid ramps at the inboard end of the dock. The sloping floor case was the same as the section of the 3-dimensional model illustrated in Figure 5.

The effect of the dock floor slope is shown in Figure 11, for both wave amplitudes. The sloping floor had a mean RAO over the wave period range that was 20% higher for the high amplitude waves and 11% higher for the low amplitude waves. Having the flooded dock floor sloping is not desirable, based on the results of the simulations.

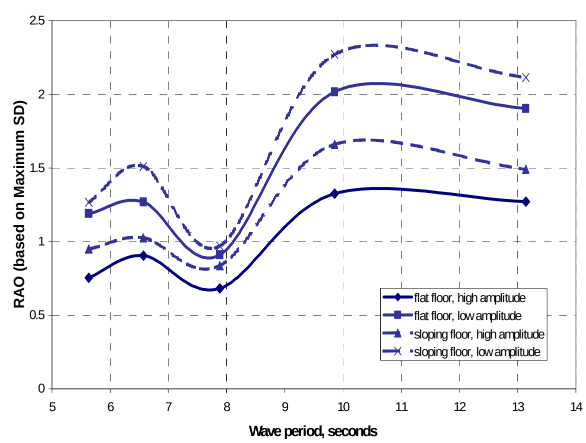

Figure 11, Effect of sloping dock floor, predicted by 2-d CFD simulations for two different wave amplitudes

4.2 b) Wave Absorption Beaches

The design of the wave absorbers at the inboard end of the well deck was also a factor of interest in the preliminary design of the well deck. Since the study was reduced to two dimensions, the most practical options to consider were sloping ramps at two levels of porosity (0 and 20%) and vertical walls, also with two levels of porosity (0 and 20%). These devices were simulated in a well deck 60 metres long, with a flat floor.

The results are shown in Figure 12 for the high amplitude waves and Figure 13 for the low amplitude waves. Table 2 gives the average RAO based on maximum standard deviation for each design, and ranks them in the order from lowest to highest.

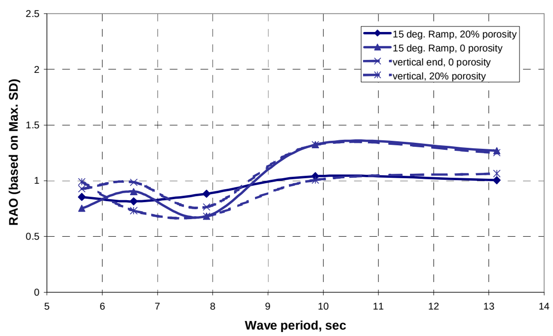

Figure 12, RAO (based on maximum standard deviation of water surface within well deck) for 0.394m wave amplitudes

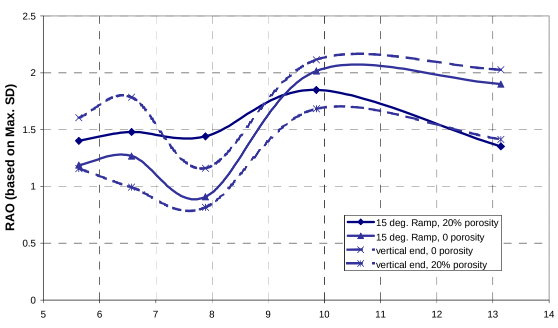

Figure 13, RAO (based on maximum standard deviation of water surface within the well deck) for 0.194m wave amplitudes

The most effective wave absorber was the vertical wall with 20% porosity, followed by the sloping ramp with 20% porosity. The difference between the two designs was of the order of 15%, averaged over the range of wave frequencies considered. This degree of porosity is effective at reducing the wave elevation within the well deck. Increasing the porosity to 30% was investigated for some situations, and it was found that the effect was almost negligible relative to 20% porosity.

End condition	Porosity, %	Ave. RAO based on max. SD 0.394m waves	0.194m waves	Average	Rank
Vertical	20	0.896	1.213	1.05	1
15 degree ramp	20	0.920	1.505	1.21	2
15 degree ramp	0	0.987	1.457	1.22	3
Vertical	0	1.051	1.738	1.39	4

Table 2, Summary of 2-dimensional simulations, effect of wave absorber design, well deck with flat floor

4.2 c) Well Deck Dimensions

When reviewing the literature on the LPDs, we found no data on the effect of dock size on wave action. Most modern LPDs have a dock area approximately 60 m long, 15 m wide and 2 m deep (See Table 1). This is sufficient for up to four landing craft to fit within the dock. However, we wanted to investigate how dock size influenced wave height. Given the geometry of a typical LPD, it would be practical to consider a dock up to 120 m long and 4 m deep. Width variation was not considered, since it had no effect on the 2-dimensional simulations.

Four dock shapes were considered, each with a flat floor and a sloping beach, with a slope of 15 degrees and zero porosity.

The results are shown plotted in Figure 14, and summarized in Table 3. The results showed that the long, shallow dock had the lowest average RAO, approximately 16 percent lower than the ‘standard’ dock. The reduction in wave amplitude was most significant at higher wave periods. Increasing water depth increased the RAO in the well deck above the standard sized dock, by over 30 percent. Water depths of 2 m are close to the draft of a typical landing craft. As the water depth is increased, it appears that the water in the dock is picking up more energy from the external waves, even though the energy in the waves decreases as a function of depth below the surface.

The results of the simulations show that a ship with a 120 m long dock would have lower average wave heights than a 60 m long dock. However, there would be a significant reduction in ‘dry’ lane-metres for land vehicle storage within the ship. It should be noted that the review of LPD dimensions given in Table 1 shows some of the ships in service do have docks approximately 120 m long. It would be interesting to compare the operational experience for these ships and a ship with a 60 m long well deck against the numerical predictions.

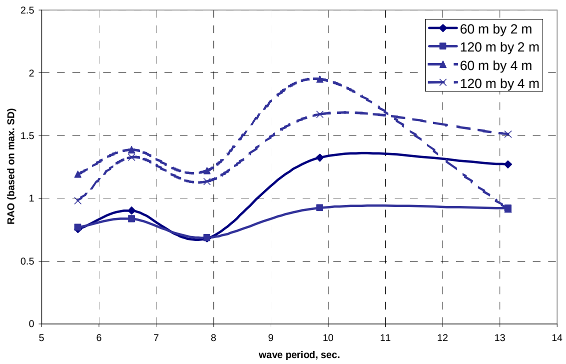

Figure 14, 2-d CFD simulations of different dock dimensions, high wave amplitude, beach with 15 degree slope, zero porosity

Length, metres	Depth, metres	Average RAO	Rank
120	2	0.829	1
60	2	0.987	2
120	4	1.326	3
60	4	1.334	4

Table 3, Effect of dock dimensions on wave amplitude in well deck, flat floor, solid ramp with 15 degree slope

5. IMPROVEMENTS REQUIRED TO NUMERICAL MODELS

The work described in this paper focused on the simplest operating scenario for an LPD, which was considered to be for the ship with zero forward speed in head waves. Whilst this is a common scenario, it is not the most complex. A good numerical procedure should be able to cope with the more complex conditions as well. In particular forward speed for the ship and additional headings should be included for the method to be complete. Preliminary studies had shown that there was potential for reasonable predictions of internal wave height for zero speed in beam waves, but no detailed data from model experiments was available with which to refine the simulations. Additional headings should be simulated in more detail and compared with measurements from model experiments.

Including forward speed within the simulations proved to be too demanding for the computer system used for this research. The most correct option of having waves in stationary water and a moving ship required a computational domain that was not practical to solve with the computer capacity available. An alternative approach of adding a current flowing past a stationary ship changed the velocity distribution within the waves, and gave unrealistic hydrodynamics. The most promising option investigated was to adjust wave period to encounter period seen by a moving ship, rather than the period seen by a stationary observer. Whilst this option gave more realistic predictions of wave amplitude in the well deck, there are other hydrodynamic features, which result from a moving ship that are not modeled. In particular, these include changes to the water level inside the well deck due to sinkage and trim caused by forward speed. Model experiment data suggests that forward speed (in head waves) lowers the wave height within the well deck, and whilst zero speed may be the most conservative, it is desirable to be able to predict the effects of ship speed on the results. This is another area that requires further development.

We feel that in order to improve the numerical methods further it will be necessary to carry out some model experiments for the specific purpose of collecting data to use for validation. The data on floating LPDs provided for the project was obtained from model experiments carried out for overall project evaluation. Whilst we felt it was extremely useful to give us checks on the results, it did not provide enough insight into the physical mechanisms involved. For example, there were no experiments in regular waves.

In order to gain the necessary insight, we recommend carrying out experiments with a model LPD in regular head and following waves, with two wave amplitudes over a range of frequencies and operational speeds for the ship (zero to six knots). It is important to understand the response of the water in the well deck to a single input frequency and to understand in detail how factors such as relative motion at the well deck entrance influence the results.

The focus of this research has been directed to validating numerical predictions of well deck performance against model data. Much of the model work has been carried out at scales of approximately 1:40. It is possible that damping devices suffer from viscous scale effects, which are not properly scaled to full scale. An important area where no data was available for validation was full-scale trials. We recommend that the numerical methods be compared to the results of full-scale trials, where ship motions and wave action inside the well deck were measured for different significant wave heights and periods, for a range of ship speeds and headings.

6. CONCLUSIONS

The primary objective of this research was to determine the accuracy of numerical predictions of wave elevation inside the well deck of an LPD. Accurate simulations will allow design alternatives to be evaluated, without the requirement for physical model experiments. The most likely operating scenario is for the ship to be head to the waves, and stopped or moving slowly ahead. Based on the results of model experiments it seems that the effect of forward speed is to lower the wave amplitudes in the well deck, so we may consider zero speed to be the most conservative condition.

It is important to obtain the level of distortion to the wave field at the stern of the ship, since this affects the wave amplitude at the entrance to the dock. The level of distortion varies with wave frequency. The reduction in wave amplitude increases as the wave period is reduced. Based on a comparison with model experiments, it seems that the level of diffraction for ship structures over a likely range of wave periods can be predicted with Flow-3D, by assuming that the ship is stationary. However, it may be possible to predict the same effects with another code (e.g. potential methods), which should compute faster than the CFD code.

Once the level of distortion is known, the waves inside the well deck can be simulated adequately by separating the well deck from the rest of the ship. The internal waves can be modelled by generating them just outside the mouth of the dock. These waves must be based on the results of the diffraction studies described above. For the particular ship case considered in this paper, there appears to be very little difference between the coupled cases and the static cases, but this should be checked against other experiment data sets before the conclusion can be generalized. It is recommended that predictions in head waves should be based on the coupled motions, which in principle allow for the movement of the fluid within the moving ship.

CFD can be used to predict general trends in performance for different well deck designs. Based on the results of the simulations, the lowest wave heights within the well deck should occur for a long, shallow dock, with a flat floor (when the ship is flooded) and a vertical beach at the inboard end of the well deck with a porosity of 20%.

At the present time, we do not recommend modelling conditions other than head waves and zero speed. The simulations showed correct trends between wave amplitudes inside the dock and external wave periods, but more refinement in the numerical model is required. We feel that it would be possible to improve the accuracy of the simulation, but do not feel that this effort is worthwhile without obtaining suitable data with which to validate the results.

7. ACKNOWLEDGEMENTS

The authors would like to thank DRDC - Atlantic, United States Coast Guard, Royal Netherlands Navy and the Institute for Ocean Technology for permission to publish the results of their research project.

8. REFERENCES

1. Hopman, H., Kapsenberg, G. and Krikke, M. ‘Design and Hydromechanic Aspects of the Amphibious Transport Vessel for the Royal Netherlands Navy’, Naval Engineers Journal, May 1994.

2. Downs, D. S. and Ellis, M. J. ‘The Royal Navy’s New Building Assault Ships Albion and Bulwark’, Warship 97, Air and Power at Sea, RINA, June 1997.

3. Hopman, H. ‘The HNLMS Rotterdam- The First RNLN LPD: How a Long Standing Requirement Became a Reality’, Warship 2000, Warships for Amphibious Operations and Mine Warfare, RINA, June 2000.

4. du Pre, A. and Wood, M. ‘The Alternative Landing Ship Logistic’ Warship 2000, Warships for Amphibious Operations and Mine Warfare, RINA, June 2000.

5. Pawlowski, J. S. and Bass, D. W. ‘Theoretical and Numerical Study of Ship Motions in Heavy Seas’, Trans. SNAME, New York, October, 1991.

6. Bass, D. and Cumming, D. ‘An Experimental and Numerical Investigation of the Effects of Water Trapped on Deck’, Proceedings STAB 2000, 7th Int. Conf. On Stability of Ships and Ocean Vehicles’, Launceston, Tasmania, 7-11 February 2000.

9. AUTHOR BIOGRAPHIES

Don Bass holds degrees in Mathematics from Hull, London and Warwick. He joined the Department of Mathematics at Memorial University in 1969 and was jointly appointed to the Faculty of Engineering in 1983. His main interests are in Ship Dynamics and Marine Hydrodynamics with particular application to small Fishing Vessels.

David Molyneux is a senior research officer at the Institute for Ocean Technology. Over the last 22 years has carried out research into a wide range of ship hydrodynamics problems including flooding of damaged ro-ro ferries in waves, high performance hulls in calm water and waves and propulsion of tugs and icebreakers.

Kevin McTaggart is Leader of the Virtual Naval Platforms Group at Defence R&D Canada – Atlantic. He has a doctorate in Civil Engineering from University of British Columbia, where he specialized in offshore hydrodynamics and probabilistic methods. He has been at DRDC Atlantic since 1989, and has been active in the areas of seakeeping, sea loads, and dynamic stability of naval vessels. His current work is focused on modelling and simulation of naval vessels in seaways.