DISCUSSIONS

35th INTERNATIONAL WORKSHOP ON WATER WAVES AND FLOATING BODIES

HOST: YONGHWAN KIM

24-27TH AUGUST, 2020

VIRTUAL WORKSHOP

Photograph: Gangneung Sea Waves of Korea, Republic

Discussion Sheet

Paper title : Heading and spreading effects on wave run-up due to tertiary interactions

Author(s): W. Zhao, P. H. Taylor, H.A. Wolgamot, C. Ouled Housseine

Question(s) / Comment(s): These are very interesting results, and I expect that the sensitivity you show to angle of incidence and 3D spreading effects will be good news for engineers.

The numerical and experimental results mostly agree rather well, but do you think that reflections from the wavemaker and side walls of the tank may have some influence here, especially around any tank eigenmode frequencies?

Asked by : Harry Bingham

Answer: Thanks for the positive comments. The reflection effect from the wavemaker and the side walls of the tank are expected to be negligible in this study.

To eliminate the concerns of tank eigenmode resonant effects, we looked at the free-surface RAOs at the gauge (marked as ×) 5.04 m away from the side of the fixed structure. The experimentally determined RAOs agree well with the linear potential flow predictions, where the tank resonance effect was not involved.

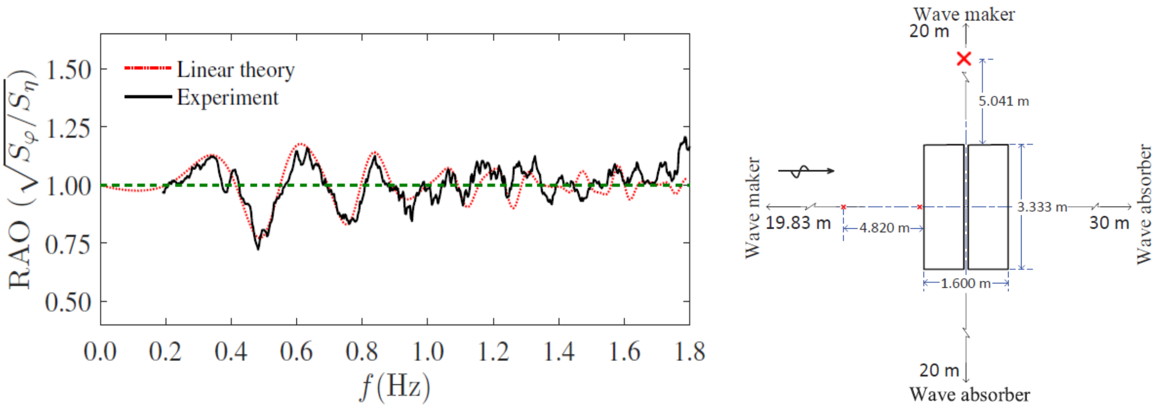


Fig 1. Numerical and exptal RAOs (left) at the side of the structure, and expt set-up (right). Incident wave energy spectrum ranges from 0.2 Hz to 1.6 Hz.

(1) Simulations: linear potential flow, no reflections & no Molin-type tertiary effects.
(2) Expts: surface elevations measured at the side gauge (red ×), with and without the structure in place, respectively.

Discussion Sheet

Paper title: A fully-nonlinear potential flow model for waterentry/exit in aircraft ditching applications
Author(s): A. Del Buono, A. Iafrati, A. Tassin, S. Ianniello

Question(s) / Comment(s):

FYI, Gravity effects on water entry/exit of a wedge were studied by two CFD codes (K. Maki and S. Seng) to include this effect into GWM and MLM, and to adopt GWM and MLM to exit problems in:

Khabakhpasheva, T.I., Korobkin, A. A., Maki, K. J., Seng, Sopheak. (2016) Water entry and exit with large displacements by simplified models. In Proc 31st International Workshop on Water Waves and Floating Bodies, Plymouth, MI, 3–6 April, 4pp.

It is hard to say that this abstract is very informative. However, the presentation (available on request) contain many figures, comparisons and results.

Asked by: A. Korobkin

Answer:
Many thanks for the suggestion. We'll ask for the presentation to do further comparisons.

Discussion Sheet

Paper title : A fully-nonlinear potential flow model for water entry/exit in aircraft ditching applications
Author(s): A. Del Buono, A. Iafrati , A. Tassin, S. Ianniello

Question(s) / Comment(s):

I note you have chosen the 2nd-order R-K scheme here for time stepping. Have you done a linear stability analysis of your discrete solution? I suspect that this problem is essentially hyperbolic, meaning that it has at least some stability eigenvalues lying on the imaginary axis. Since the R-K 2 scheme does not include any part of the imaginary axis, I suspect that this is not a good choice. You may find that less (or no) filtering may be required if you choose R-K 3 or R-K 4 for example, both of which include a large part of the imaginary axis in their stability regions.

Asked by : Harry Bingham

Answer:

Many thanks for your suggestion and we'll certainly consider the use of the fourth order RK in future.

By looking at Longuet-Higgins and Cokelet (1976) and Dold (1992), it seems that the sawtooth instability is more related to the spatial discretization rather than to the time integration and also for the present simulations, no significant changes have been found by reducing the time step. Maybe, the problem could be attenuated by using higher order representation for the velocity potential along the panels (e.g. linear distribution rather than piecewise constant).

The fourth order RK is already implemented and used in the FNL-BEM model adopted for the study of free surface waves (e.g. Iafrati et al., JCP 2014) but for the water entry/exit code, only RK2 is available at present. The reason is that in water entry/exit problems there are quite rapid changes in the free surface curvature. For this reason, the free surface discretization is updated at each time step in order to refine the panel size in highly curved region and in order to assure a satisfactory accuracy the time step has to be kept as minimum as possible. In this sense, the fourth order RK could not be used to increase the time step but only at improving the accuracy of the simulation. Owing to the doubling in the number of substeps, the computational cost (which is already quite high due to the need to rebuild the matrix of the influence coefficients at each substep) also doubles.

Discussion Sheet

Paper title : Experimental study of phase transition in sloshing-induced impact in two-dimensional tank
Author(s): J. Lee, Y. Ahn, J. Kim, Y. Kim

Question(s) / Comment(s):

In your presentation, but not in your abstract, you cite Kim et al (2017). Could you please give complete reference to this paper and give more details what has been done there?

The Conclusion in your abstract is made of three points. The first one is about observation from your experiments. The second and third points are:
The oscillation is reduced due to the phase transition in the impact with gas pocket near the boiling point.
The phase transition reduces the maximum impact pressure at the boiling point.
How do you know about the phase transition? I did not understand from the presentation that you measured mass of water, which was turned to the vapour during an impact or vice versa.

I assume that "phase transition" and reduction in the impact pressure depend not only on the type of impact but also on impact duration, which highly depends on scale of your model. Is it right that phase transition depends on impact speed, say?

Asked by : A. Korobkin

Answer:

1. The title of paper by Kim et al (2017) is "Experimental Observation of the Effects of Liquid Temperature and Bubbles on Impact Pressure Inside Gas Pocket". This study focused on the physical phenomenon of phase transition, and measured the impact pressure on the upper surface of the cylinder by dropping a hollow cylinder-shaped dropping object into boiling water. Since the inside of the cylinder is hollow, water vapor is included in the cylinder, and the effect of the phase transition of vapor on the impact pressure was investigated. The results showed that the magnitude of the impact pressure decreased, and the oscillation caused by the gas pocket decreased. However, since the flow inside the cylinder could not be observed directly, only changes through the pressure time series could be identified.

2. In this study, the phase transition could be confirmed in the following steps: First, in the

flip-through impact experiment at low filling condition, it was found through images and pressure time series that when an external impact was applied in a saturated heating condition where a phase transition could occur, the oscillation of the bubble was reduced compared to the room temperature condition. This is a phenomenon that occurs when some of the gas inside the gas pocket is liquefied due to the phase transition and loses the compressibility of gas pocket. Through this, it can be inferred that a phase transition occurred inside the bubble.
Next, under high filling conditions, it is possible to confirm the liquid liquefied by external impact pressure inside the gas pocket through an impact experiment involving a large gas pocket, which only occurred under boiling condition. In particular, since the wall surface corresponding to the thermal boundary has a greater temperature gradient than the other boundary, liquefaction may more easily occur at the wall surface having a lower saturation pressure than the surrounding when the same pressure is applied in the saturation state.

3. In order to examine the effect of the phase change phenomenon, the same Froude scaling was applied to the size of the tank and the size of motion under the conditions of room temperature water, NOVEC 7000 at room temperature, and NOVEC 7000 at boiling point. In addition, in the case of boiling conditions of NOVEC 7000, similar to the actual cargo, it has a cavitation number of 0. In order to examine the dependence of the phase transition through the cavitation number under different temperature conditions, velocity of fluid is required.
However, since this study focused on the effect of phase transition on the impact pressure, the exact velocity of the fluid was not measured. The relationship between the impact velocity and various physical phenomena including phase transition will be analyzed by measuring the accurate flow velocity through PIV experiments in the future.

Discussion Sheet

Paper title : Ship-motion Green function with viscous effect
Author(s): Xiaobo Chen, Hui Liang and Young-Myung Choi

Question(s) / Comment(s): Thanks for a very interesting talk. You closed by asking the question: "What value of viscosity should be chosen?". Why is the answer not simply the true fluid viscosity?

Asked by : Harry Bingham

Answer:

Yes, we have started with the use of the "true" (molecular) fluid kinematic viscosity $\nu = 1.15 \times 10^{-6}$ as involved in (6) to define the parameter ϵ. Indeed, the so-defined ϵ =1/Re is the reciprocal of Reynolds number if the length is taken as U^2/g. The value of ϵ is of order 10^{-5} to 10^{-8} for U from 1 *m/s* to 10 *m/s*. In reality, the dissipation could be more important than that relative to the molecular viscosity due to many physical phenomena like effect of wave breaking, wind air friction, turbulence, variation of surface tension, etc. We take often the value 10^{-4} for ϵ to have wave patterns without highly oscillatory short waves.

Discussion Sheet

Paper title : A coupling of linear potential flow model based on Poincar´e's velocity representation and viscous flow model based on SWENSE
Author(s): Young-Myung Choi, Benjamin Bouscasse, Lionel Gentaz, Pierre Ferrant, and ˇSime Malenica

Question(s) / Comment(s):

In the potential flow zone, the incident wave is modeled by a fully nonlinear model and the complementary component is computed/represented by a linear model, does the total wave free surface satisfy the nonlinear free surface boundary conditions or linear ones?

Asked by : Xingya Feng (SUSTech)

Answer:

When the radiation-diffraction waves is small in the far-field and the influence from the far-field solution to the near-field is small, it can be said that the nonlinear free surface boundary condition is satisfied. In the vicinity of the body, the fully nonlinear models both for incident and radiation-diffraction waves are considered, e.g. nonlinear condition is satisfied. If the radiation-diffraction waves are small in the far-field, the free surface condition can be approximated into the linear case. For a general 3D case, the radiation-diffraction waves decays with O(1/r), the assumption we introduced seems appropriate.

However, the radiation-diffraction waves are significant in the far-field, it is hard to say that the proposed method satisfies fully nonlinear and linear conditions on the free surface. The results with diffraction by a cylinder are compared with results from the fully nonlinear Navier-Stokes viscous flow solver in a large computational domain, the proposed method shows similar wave patterns.

In addition to the wave amplitude, we can think of the effect of viscosity that can survive up to the far-field. Then, it can be also difficult to say that the free-surface boundary condition satisfied both in viscous and potential flow domain.

Discussion Sheet

Paper title : CFD based detection of slamming loads for calibration of a pressure-impulse model

Author(s): A. Ghadirian, H. Bredmose

Discusser: Yonghwan Kim

Question(s) / Comment(s):

Q1. You mentioned about pressure impulse method. However, you described the results with only forces. The force is an integrated value of pressure distribution. So, could you confirm if you want predict pressure or force?

Q2. According to your force time series of 5 cases, the impact occurs at different times in case (c) and (e), and OCW3D didn't provide significant impulsive signals for case (a), (b), and (d). But all the signals of OpenFOAM results show impulse occurrences in all the cases. I assume that the flow patterns may be very different in the two different methods? Do you have any observation and comparison on the detailed wave and flow patterns?

Answer:
Thank you very much for your questions.

A1. The model is a solution to the boundary value problem of the pressure impulse and also gives pressure impulse as output. In this abstract, we integrated the pressure impulse on the surface of the cylinder and calculated the force impulse. The choice of comparing the force impulses instead of pressure impulse distributions was merely for illustration and brevity purposes. In future more comparisons of the pressure impulse distributions will be conducted.

A2. Yes, the flow patterns are sometimes, especially for large breaking cases different in OpenFOAM and Oceanwave3D solvers. TOceanwave3d is a potential flow solver and includes only a smearing function to dampen the waves when a wave is "supposed" to break. OpenFOAM on the other hand is a VOF Navier-Stokes solver which can solve wave episodes that are breaking. In addition, Oceanwave3d does not include the structure so the scattered waves are also not present there. So, the differences to some extent were expected and actually welcome because ideally, we wanted all of the cases to look like case (a) where the

Oceanwave3d solution is smooth but the OpenFOAM has one abrupt increase in the inline force time series. But the large differences were there because of the mapping and other numerical problems. Please note that we were trying to create these breaking waves to break (or slam) at the cylinder on a flat bed without directional spreading.

Discussion Sheet

Paper title : Singularity in the second order flexural-gravity waves

Author(s): Teng B., Liang S., Gou Y., Korobkin A.A., Malenica S.

Question(s) / Comment(s):

Figures 8 and 9 show that the second order force tends to infinity. Doesnt this invalidate the assumption of a regular perturbation analysis where each term is smaller than the previous one?
Is there any experimental evidence to confirm this behaviour?

Asked by : David Evans, University of Bristol

Answer:

For a truck running on an ice sheet, there is a speci al speed at which the ice sheet will vibrate greatly. For the present problem I am not sure if it is a phy sical problem or a numerical problem with the pertu rbation method.

We are developing another model for second order a ction with a plate with a finite length in which the second order flexural-gravity wave is not used. If the similar phenomenon can be found, we will try to carry out model test to confirm it.

Thanks for your suggestion.

Discussion Sheet

Paper title : Singularity in the second order flexural-gravity waves
Author(s): Teng B. et al.

Question(s) / Comment(s):

When we consider the flexural-gravity waves, the extra wavenumbers locate in the complex domain in my understanding. These complex eigenvalues (wavenumbers) may affect the eigen-functions and the number of modes to be considered.

Here are the questions:

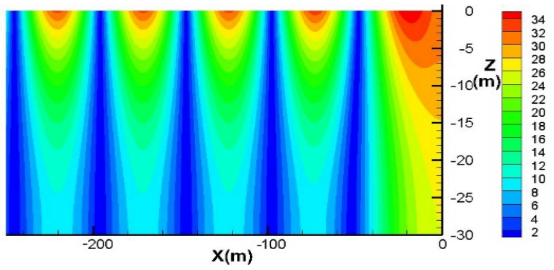
1. The effect of complex wavenumbers on eigen-functions including the extra wavenumbers.
2. How much extra modes affect the solution and the possibility of other wave system due to extra wavenumbers.

Asked by : Young-Myung Choi (Bureau Veritas)

Answer:

Yes, you are right. We have included those evanescent modes due to imaginary roo ts and complex roots of the dispersive equation in the reflection waves, and found that the singularity phenomenon is not from them.

We included enough evanescent modes to guarantee the results are converged. The effect of those evanescent modes is at the nearby of the vertical wall.



Discussion Sheet

Paper title : On the generalized motions/deformations of the floating bodies
Author(s): Malenica S. & Bigot F.

Question(s) / Comment(s): Thank you for this important result which seems to elegantly settle a confusing issue that has been discussed for many years in the hydrodynamics community. It seems that the new terms that come out of this analysis only involve cross-coupling modes between the generalized modes and the rigid-body modes, is that correct?

Asked by : Harry Bingham

Answer: That is correct. The difference with the previous formulations of the hydrostatic restoring matrix, is related to the cross coupling terms between the flexible (generalized) modes and the rigid body modes. This is due to the fact that the dynamic motion equation is written in the body fixed coordinate system where the change of the normal vector and the mode shape vector are induced by the flexible motions only! At the same time the gravity vector changes when expressed in the body fixed coordinate system. These are two fundamental points which should be taken into account very carefully.

Discussion Sheet

Paper title :
Natural modes in three-dimensional rectangular moonpools with recess in finite-depth waters

Author(s):
X. Zhang, Z. Li, J. Han

Question(s) / Comment(s):

There is an issue concerning the expansion in the subdomain (3) based on the zero-potential on boundaries T1-T4 without coupling with the potential in the outer domain. The work by N. B. Dişibüyük; A. A. Korobkin; and O. Yilmaz about "Linear Wave Interaction with a Vertical Cylinder of Arbitrary Cross Section: An Asymptotic Approach" (Journal of Waterway, Port, Coastal, and Ocean Engineering, 143 (5) 2017), could be useful.

How about natural modes of circular moonpool with recess in a circular cylindrical structure? There is a consistent solution based on eigen-expansions in different subdomains as presented in Chen, Liu & Duan "Semi-analytical solutions to wave diffraction of cylindrical structures with a moonpool with a restricted entrance" *J Eng Math* 90, 51–66 (2015).

Asked by : Xiaobo CHEN

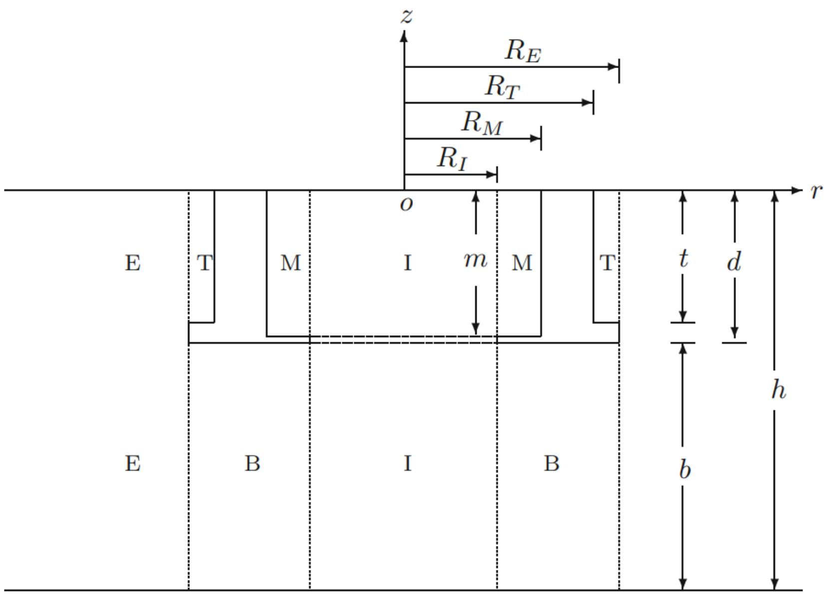
Answer:

Thank you very much for your comments and suggestions.

Regarding your first comment, we assume the velocity potential is zero at the outer boundaries, and develop the model without considering the potential in the outer domain. Based on our results, we found that this simplification does not obviously impact the prediction of the natural frequencies of the moonpools.

The suggested reference investigates on a bottom-mounted vertical cylinder. Asymptotic expansion by Taylor formula is applied on the solution of velocity potential of the whole fluid domain. The radiation problems are solved by the Fourier method. It seems that paper focused on solving the problem for bottom-mounted vertical cylinders with arbitrary cross-section.

Regarding the second comment, we appreciate you provide the valuable reference. We actually are working on the case for circular moonpools with recess. We have compared the solutions using our three-dimensional model with your results and the measurements. The table below shows the variation of piston mode frequency with the recess radius. As shown, those from solving the eigenvalue problem and frozen mode approximation model are in satisfactory agreement with the experimental data.



R_I(m)	23.5	29	32	34.5
FMA ($\mathbf{rad\ s^{-1}}$)	0.3519	0.3900	0.4068	0.4177
Eigenvalue problem ($\mathbf{rad\ s^{-1}}$)	0.3636	0.3999	0.4144	0.4217
Chen 2015 ($\mathbf{rad\ s^{-1}}$)	0.3510	0.3831	0.3977	0.4028
Exp. ($\mathbf{rad\ s^{-1}}$)	0.3491	0.3740	0.4002	0.4053

Discussion Sheet

Paper title :
Prediction of Wave Loads with Measured Unsteady Pressure Distribution on Ship-Hull Surface

Author(s):
Masashi Kashiwagi, Hidetsugu Iwashita, Kurniawan T. Waskito and Munehiko Hinatsu

Question(s) / Comment(s):

Asked by : Harry BINGHAM

Congratulations on this very impressive set of experiments! Can you say something about the computational time required to make those calculations?

Answer: by Masashi KASHIWAGI

If you are asking the CPU time for CFD computations uing FINE/Marine for this problem, it takes approximately 4 days per one wavelength, in which the computations are made for 15 to 20 wave periods after attaining the steady forward speed and oscillation (in addition to 10 to 15 wave periods for acceleration starting from the state of rest).

Discussion Sheet

Paper title :
Prediction of Wave Loads with Measured Unsteady Pressure Distribution on Ship-Hull Surface

Author(s):
M. Kashiwagi, H. Iwashita, K.T. Waskito, M. Hinatsu

Question(s) / Comment(s):

Asked by : Xiaobo CHEN

Excellent experimental work! Your new series of model test results should be very useful in benchmark comparison with numerical methods. I wonder if any chance/way to get the geometry data and measurements for comparison with BV numerical tools.

Answer: by Masashi KASHIWAGI

The RIOS bulk carrier was made for academic purpose with intention of disclosing the offset data to the public. In fact, we have already provided the geometric data of the ship to some organizations.

Regarding measured data, we have not only the spatial distribution of unsteady pressure, but also hydrodynamic forces (including added resistance), wave-induced ship motions, unsteady wave patterns; all of these have been measured as validation data to be made open to the public. However, especially on the pressure distribution, we are going to write a journal paper regarding some details in the experiment, particularly effects of various factors influencing the accuracy and reliability of the data, such as the temperature variation, position of air-water interface, surface tension, and so on. Once we could write that journal paper, we will surely open the data to the public.

Discussion Sheet

Paper title : Prediction of Wave Loads with Measured Unsteady Pressure Distribution on Ship-Hull Surface
Author(s): M. Kashiwagi, H. Iwashita, K.T. Waskito, M. Hinatsu

Question(s) / Comment(s):

Many thanks for such an interesting study. I believe the technique you're employing is really powerful and may help the development and the validation of numerical approaches, as well as the understanding of the physics. Concerning the experimental methodology, in slide 6 of your presentation, you discuss the general approach to the VBM estimates, which is usually based on segmented hull concept. The use of pressure probes indeed allows a much finer representation of the loading. However, segmented hulls are also used to reproduce the actual ship stiffness. It seems to me that in your experiments the model is not segmented and I'm wondering if the stiffness of the model has been scaled properly or if it is just a rigid model.

Asked by : Alessandro Iafrati

Answer: by Masashi Kashiwagi

You noticed very important point. Our ship model is basically rigid, and thus we have assumed the weight distribution. If we will use a model with actual ship stiffness, the calibration for the FBG pressure sensor may be problematic. Thus enhancement of reliability of the sensor (such that there is no need to repeat the calibration) will be a key for a future application.

Discussion Sheet

Paper title :

Numerical Simulation of Air Cavity with Water Waves

Author(s):

Yuxi Huang, Xin Wang, Arun Dev, Dominic A. Hudson

Question(s) / Comment(s):

Asked by : Masashi Kashiwagi

(1) Looking at Table 5, the wave-amplitude dependency on the drag reduction looks different between Fr=0.1949 and Fr=0.2274. What do you think is a reason of this difference?

(2) In actual seas, the added resistance due to wave making would be larger than the reduction of skin friction resistance by Air Lubrication System. Do you have some information on the relative magnitude?

Answer:

Thank you for the questions.

(1)

For easy reference, Table 5 of wave-amplitude dependency is shown below:

Freestream Velocity (Fr)	Wave	Air Flux (m^3s^{-1})	Min. Sweep Angle	Max. Sweep Angle	Mean Sweep Angle	% Drag Reduction
$1.647ms^{-1}$ (0.1949)	Calm	2×10^{-3}	-	-	75.13°	5.10%
		2.5×10^{-3}	-	-	73.03°	4.40%
	$H_W =$ 0.6m	2×10^{-3}	71.66°	76.47°	74.37°	9.96%
		2.5×10^{-3}	71.73°	73.47°	72.48°	9.89%
	$H_W =$ 0.12m	2×10^{-3}	72.93°	74.96°	74.10°	11.74%
		2.5×10^{-3}	72.12°	75.66°	73.76°	11.58%
$1.922ms^{-1}$ (0.2274)	Calm	2×10^{-3}	-	-	78.32°	4.91%
		2.5×10^{-3}	-	-	78.30°	2.61%
	$H_W =$ 0.6m	2×10^{-3}	76.91°	80.56°	78.18°	11.76%
		2.5×10^{-3}	75.71°	77.34°	76.60°	11.08%
	$H_W =$ 0.12m	2×10^{-3}	77.96°	79.35°	78.70°	4.58%
		2.5×10^{-3}	77.10°	78.49°	77.95°	4.03%
$2.196ms^{-1}$ (0.2599)	Calm	2×10^{-3}	-	-	80.05°	3.50%
		2.5×10^{-3}	-	-	79.54°	3.11%
	$H_W =$ 0.6m	2×10^{-3}	77.75°	81.84°	80.28°	3.74%
		2.5×10^{-3}	78.21°	80.86°	79.29°	3.43%
	$H_W =$ 0.12m	2×10^{-3}	78.32°	81.42°	79.48°	6.3%
		2.5×10^{-3}	77.36°	80.46°	78.62°	5.75%

Table 5: Different observed sweep angles and drag reduction

Wave dependency patterns are different for Fr=0.2274 in that a larger wave height resulted in a lower drag reduction whereas other cases, the opposite is true.

Drag reduction is dependent on flow characteristics in boundary layer; a major one being impingement of air. An increase in volume fraction of air (impinged) will lower wall shear stress. It can thus be assumed impingement can have a large effect on drag reduction.

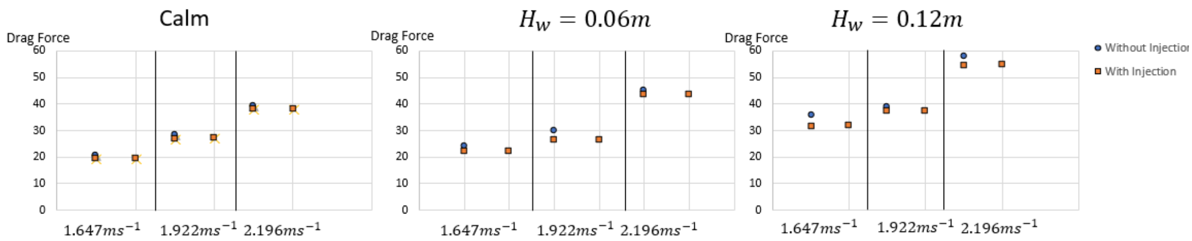
Impingement behaviors is ultimately a balance of the following three items:

- Momentum from injection
- Freestream velocity (cross flow parallel to ship hull)
- Buoyancy

In frame of reference for these simulation, injection and free stream velocity is ever changing due to ship movement. Drag reduction observed may be the manifestation of less than ideal impingement based on specific flow conditions.

(2)

Indeed, in actual seas, the added resistance from waves will be much larger than reduction of skin friction. The three plots show the absolute magnitude of drag force experienced by ship model.



In terms of percentage change:

	% Increase in Drag (No Injection)	
Freestream Speed (Fr)	$H_W = 0.06m$	$H_W = 0.12m$
$1.647ms^{-1}$ (0.1949)	18.27%	74.54%
$1.922ms^{-1}$ (0.2274)	5.43%	38.39%
$2.196ms^{-1}$ (0.2599)	14.3%	47.81%

As mentioned in the abstract, a larger drag reduction seen in waves is by no means prove that sailing through rough seas makes air lubrication systems more effective as added wave resistance exceedingly overwhelms what drag reduction with ALS can provide.

Discussion Sheet

Paper title : A note on a trial to improve linearized solution in water wave problems
Author(s): I. J. Lee, E. S. Kim, S. H. Kwon

Question(s) / Comment(s):

1. **I think RAO is independent of analysis methods. Why the RAO's from the new method are much different from those from classical methods? Have y ou validated your new results against model test data?**

Asked by :

Answer:

Thank you for your question.

First, the present study is different with analysis methods. We have reformulated most existing equations, from BVPs to EOMs, by introducing the fluid displacement, instead of using structure displacement. As a result, new fluid force and moment are derived. As can be seen, both the force and moment have quite different forms with those from classical approach. If we decompose them into scattering and radiation components, we can obtain new equation of motions, and it leads us to get new RAOs.

Second, we have not yet validated them with experimental measurements. We just have compared them with numerical results from classical approach.

Discussion Sheet

Paper title : A note on a trial to improve linearized solution in water wave problems
Author(s): Ik Jae Lee, Eun Soo Kim, Sun Hong Kwon

Question(s) / Comment(s): This is an interesting idea, but I'm not sure that you have convinced me that you have improved on the traditional linearized solution. For the response of a heaving and surging hemisphere for example, the traditional solution compares quite well with experimental measurements for small amplitude motions, yet your new solution is very different. What evidence do you have to support the claim that the new solution is an improvement?

Asked by : Harry Bingham

Answer:

First of all, we are sorry that the title might give you some confusion as if we have already made improvement. Satisfactory results have not yet been achieved. We would appreciate if you consider a point that we are at the very beginning of this new idea.

In our pre-recorded presentation, we addressed one possible error source which we called geometric singularity. Specifically, we cannot take the Taylors series expansion in the vicinity of sharp corner and edges, because the location of position vectors after deformation can be slipped out of the surface, if we take the fluid displacement. That is the point we have presented on the recorded video. However, later we figured out that even without geometric singularity the location of position vectors may not remain on the body surface if the surface is curved. That is, only over the flat surface, the concept of fluid displacement is perfectly applicable because the position vector can remain on the body surface even after its tangential movement. In our conjecture, the case of heaving hemisphere you mentioned is something like that of curved surface.

Second, at some cases, the RAOs are well-matched with experimental measurements, but not every case. It is a well-known fact that without artificial damping coefficient, RAOs from traditional method can have unusual high resonance region. But as you can see in the result of circular cylinder, the resonance region disappears. This is because in our new approach, there are newly appearing complex-valued terms from the fluid displacement, and they play additional damping roles in the equation of motions. And these are not artificial, but physical.

Discussion Sheet

Paper title : Extraction of higher harmonic wave elevation and loads using a four-phase approach in fully nonlinear simulations
Author(s): Xingya Feng, Paul H. Taylor, Wei Bai, Thomas A. A. Adcock

Question(s) / Comment(s): Could you explain how you compute the moment arms for each order?

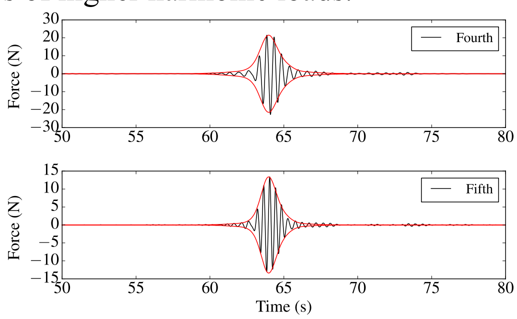
Asked by : Harry Bingham

Answer: by Xingya

We first compute the envelopes of the force and moment at each order from the time series.
The envelopes can be simply obtained by Hilbert transform.
The arm at each order is then defined by the peak moment over peak force as:

$$L_n = \frac{\max(\sqrt{M_n^2 + M_{nH}^2})}{\max(\sqrt{F_n^2 + F_{nH}^2})}$$

Examples of the envelopes of higher harmonic loads.



Discussion Sheet

Paper title : On mode-1 and mode-2 internal solitary waves in a three-layer fluid system

Author(s): T.Y. Zhang, Z. Wang, Z.H. Wang, B.T. Xie, B.B. Zhao, W.Y. Duan, M. Hayatdavoodi

Discusser: Korobkin

Question(s) / Comment(s):

I assume that one of the aims of your study is to confirm the HLGN model by experiments. Any other aims?

Answer:

Thank you for your question.
Your question is related to our future research.
Our target is use N-layer HLGN model to simulate the real-scale internal waves in the South China Sea.
So, present 3-layer HLGN model and the validation helps the N-layer HLGN model validations and applications.

The N-layer HLGN model is similar with the N-layer models developed by Liu&Wang(2012, JFM). In their paper, they write "However, the model is limited by the assumption that the total water depth is shallow in comparison with the wavelength of interest. Furthermore, the vertical vorticity must vanish, while the horizontal vorticity components are weak."
But the N-layer HLGN models has no such restriction.
So, we want to develop the N-layer HLGN models.

Discussion Sheet

Paper title : On mode-1 and mode-2 internal solitary waves in a three-layer fluid system

Author(s): T.Y. Zhang, Z. Wang, Z.H. Wang, B.T. Xie, B.B. Zhao, W.Y. Duan, M. Hayatdavoodi

---

Discusser: John Grue

Question(s) / Comment(s):

A comment to the efforts on mode-2 waves, which is undergoing active research right now. Here are some references that are relevant:

D. Deepwell, M. Stastna, M. Carr, P.A. Davies. Interaction of a mode-2 internal solitary wave with narrow isolated topography. Phys. Fluids 29,076601, 2017.

D. Deepwell, M. Stastna, M. Carr, P.A. Davies. Wave generation through the interaction of a mode-2 internal solitary wave and a broad isolated ridge. Phys. Rev. Fluids 4, 094802, 2019.

M. Carr, M. Stastna, P.A. Davies, K.J van de Wal. Shoaling mode-2 internal solitary-like waves. J. Fluid Mech. 879, pp. 604-632, 2019.

You may also check the references therein.

Answer:
Thank you for your comments and very helpful references.

Discussion Sheet

Paper title : On drift motion of deformable ice sheets by nonlinear waves
Author(s): Vasily Kostikov, Masoud Hayatdavoodi, R. Cengiz Ertekin

Question(s) / Comment(s):

The equation of drift force includes two terms, the water-line integration of wave r un-up and body surface integration of pressure. 1. Is the water-line integration can be neglected for this problem? 2. How is the draft of the plate is deleted from the body surface integration at two ends of a plate?

Asked by : Bin Teng

Answer:

Thank you for such a good questions! In our approach we divide the flow domain into two types of region: open water region and region under the plate. The contribution of pressure on the side walls of the plate where the water-line is present can be neglected in our model, because we consider the thin plate approximation. This way, the total horizontal force is calculated by integration of fluid pressure along the bottom surface of the plate only. This is also an answer to the second question. Note, that in our scheme we account for non-zero draft of the plate, making the thickness of the fluid sheet under the plate smaller than in the open water region according to the plate mass and buoyancy principle.

Discussion Sheet

Paper title : Flexural Gravity Wave Scattering for Compressed Ice
Author(s): M. H. Meylan, T Sahoo, S.Das

Question(s) / Comment(s):

1. What is the physical source for the compressive stress?

2. You say that energy is conserved despite R and T sometimes exceeding unity. It ought to be possible to derive an expression for energy conservation involving the magnitude of both R and T which would also then provide a check on your numerical work.

Asked by : David Evans University of Bristol

Answer:

1) **Currents or wind can produce significant compression in the ice. Perhaps the most famous example is the compression which sent the Endurance of Sir Ernest Shackleton 1914 – 1917 Trans-Antarctic expedition to the bottom of the Weddell Sea. We also know that the compression routinely exceeds the buckling strength of the ice and forms large pressure ridges.**
2) **Yes – the conservation of energy identity has been derived (for scattering by a crack) and appears in a paper under review. It would be interesting to extend this identity to this case and the three-dimensional case.**

Discussion Sheet

Paper title : Flexural Gravity Wave Scattering for Compressed Ice
Author(s): M. H. Meylan, T. Sahoo, and S. Das

Question(s) / Comment(s):

Once the compression becomes large enough, two more real roots of the dispersion equation for certain values of wave frequencies are excited compared to the case with Q = 0 (i.e., no expression). The origin of these extra solutions is believed to be the complex roots of the dispersion equation which occur when Q = 0. At the critical point where the roots first become real, there is "a double root" on the real axis. Does that mean the original two complex roots for Q = 0 merge into one single real root? For this critical situation, will singularity occur in the eigenfunction matching process solving the unknow coefficients?

Asked by: Siming Zheng (University of Plymouth)

Answer: It turns out that when the roots merge, we chose one to be positive and one to be negative, so we do not actually have double roots. The reasons for this choice are far from obvious, and we debated this for some time. A detailed discussion is given in a paper currently under review.

Discussion Sheet

Paper title : Numerical study towards closed fish farms in waves using two Harmonic Polynomial Cell methods
Author(s): Yugao Shen, Marilena Greco, Odd M. Faltinsen, Shaojun Ma

Question(s) / Comment(s): In your last example, it looks like there might in reality be wave breaking. Have you introduced a breaking model or any filtering into your calculations?

Asked by : Harry Bingham

Answer:

Thank you for the question. **No breaking model** was introduced. Instead, a **savitzky-golay filter** was implemented in the calculation. Also, to avoid possible **plunging wave reentering the free surface**, which will lead to the breakdown of the potential flow solver, the markers with the **radius of the curvature** less than a given value, on the free surface, **were removed** every time step.

Actually, for cases with intermediate internal water depth, breaking wave was observed in the model tests and will be examined in the future.

Discussion Sheet

Paper title : A modified Benjamin-Feir index for crossing sea states

Author(s): Shuai Liu, Xinshu Zhang

Discusser: Korobkin

Question(s) / Comment(s):

Could you please say more about the conditions under which interaction of two waves described by equations (2) and (3) from your abstract can be reduced to a single equation (7), which describes time-evolution of a fictitious wave?

Is it right that \alpha in (4) is proportional to the frequency?
In
Karjanto, N. (2019). The nonlinear Schrodinger equation: A mathematical model with its wide-ranging applications. arXiv preprint arXiv:1912.10683.
a similar coefficient is proportional to the second derivative of the frequency with respect to the wave number.

Answer:

Here, we considered an ideal condition, two identical components with different direction. We assume that the evolutions of theses two wave system are the same. I feel like that is rather limited and cannot be used for crossing seas in general. My start point here is to study the effect of crossing angle. It is just the first step and recently I am trying to include the effects of frequency difference, wave height difference and so on.

Yes, \alpha in (4) is proportional to the frequency. I am not sure whether this CBFI is similar to Karjanto's. I will read this paper carefully to find out. Thank you for the inform ation.