VII International Conference on Computational Methods in Marine Engineering
MARINE 2017
M. Visonneau, P. Queutey and D. Le Touzé (Eds)

Estimation of Added Wave Resistance with CFD Code

Romain Huret*, Doriane Causeur†, Anne-Sophie Dubois‡, Aurélien Drouet†, Olivia Thilleul°

* DCNS Research/SIREHNA
Technocampus OCEAN
5 rue de l'Halbrane, 44340 Bouguenais, France
Email: romain.huret@sirehna.com - Web page: http://www.sirehna.com

† HYDROCEAN
8 Boulevard Albert Einstein, 44000 Nantes, France - Web page : http://www.hydrocean.fr
Email: doriane.causeur@hydrocean.fr, aurelien.drouet@hydrocean.fr

‡ STX France SA
Avenue Bourdelle CS 90180, 44613 Saint-Nazaire, France
Email: anne-sophie.dubois @stxeurope.com - Web page: http://www. stxeurope.com

° IRT Jules Verne
Chemin du Chaffault, 44340 Bouguenais, France
Email: olivia.thilleul@irt-jules-verne.fr – Web page: http://www.irt-jules-verne.fr/

Key words: Computational Methods, Wave, Sea Keeping

This paper is part of the Bassin Numérique project managed by IRT Jules Verne (French Institute in Research and Technology in Advanced Manufacturing Technologies for Composite, Metallic and Hybrid Structures). The authors wish to associate the industrial and academic partners of this project; respectively DCNS Research/SIREHNA, HYDROCEAN, STX France, Bureau Veritas.

Abstract. A numerical towing tank needs to efficiently estimate the ship performances in both calm water and in regular waves. The knowledge of ship performances are mandatory during the design phases in order to provide architects with values helping in technological choices. In this context, numerical towing tanks appear as a more versatile solution than time consuming and costly model tests. The French Technical Research Institute IRT Jules Verne conducts studies to assess and validate methodologies based on CFD simulations to evaluate added resistance in regular waves.

The present work conducted in the "Bassin Numérique" project provides a preliminary sensitivity analysis which aims to validate the numerical settings necessary to model the wave propagation. The main result of this preliminary study enables to specify accurate meshes for wave propagation.

The present paper focuses on the validation study done by three different members of the IRT Jules Verne using three CFD solvers on four test cases: one static vertical cylinder [1] and three ships in head wave condition [2], [3] and [4]. For each case, numerical results are compared with towing tank experiments in terms of added resistance and motions.

The different wave conditions and test cases allow covering the wide range of encountered

wave frequencies and dealing separately with the cases of diffraction at zero speed, diffraction with forward speed and finally including radiation. Most of the results correctly fit the experimental data, especially in terms of heave and pitch. The added resistance is also accurately simulated for sufficiently high wave lengths.

1 FLOW SOLVERS AND MESHES

For the purpose of the study, three flow solvers are used to investigate different tests cases. These three solvers are STAR-CCM+, ISIS-CFD, NavalFOAM. They solve the RANSE (Reynolds Average Navier-Stokes Equations) by mean of Finite Volume methods. These solvers allow setting simulations in various ways for examples by choosing the linear solver algorithms the discretization schemes… The main settings used are briefly presented below.

The boundary conditions used are:

- Inlet velocity, or wave generation condition upstream,
- Zero pressure gradients downstream
- Slip wall on the bottom of the domain
- Hydrostatic pressure on the top of the domain
- Symmetry condition on the side of the domain
- No slip, wall function on the ship

Turbulence is modeled with the common two equations eddy-viscosity formulations k-ω SST. This closure model predicts the turbulence by means of the kinematic turbulent intensity (k) and the turbulent specific dissipation rate (ω). Wall functions are applied on the ship hull except deck which is almost always set as a slip wall (i.e. no friction).

Because of different implementations in the solvers, the wave is either a 2nd order Stokes wave (ISIS-CFD) or a 5th order Stokes wave (STAR-CCM+ and NavalFOAM). The free-surface is modeled by the tracking method VoF (Volume of Fluid). With this method waves can be reflected by the outlet back in the domain. This automatically leads to wrong and non-converged solution. To overcome this problem the same kind solution is used with all solvers. It consists in adding a damping source term in the momentum equation to reduce to zero the vertical component of the velocity due to the wave at the outlet. The zero gradient boundary condition can therefore be respected at the outlet. In case of free trim and sinkage simulation, the ship motion is taken into account by deforming the mesh using the morpher included in the solvers.

The meshes are generated by the recommended tools of each solver: the integrated mesh generator is used for STAR-CCM+, Hexpress provided by Numeca is used for ISIS-CFD; snappyHexMesh is used for NavalFOAM.

2 TEST CASES

The four test cases used in this study have been chosen for their wide range of sea states (Table 1) and because of the different experimental data available.

The first test case is a fixed cylinder in regular wave [1]. This is an academic case with experimental data from the wave basin of ECN (Ecole Centrale de Nantes). The waves used for this case have the greatest camber among the entire waves used in this study. A mesh and time step sensitivity analysis is realized for this set up, the final results are presented in next section. For this diffraction case the water elevation is measured experimentally at three positions around the cylinder, and the first three harmonics of drag force are also available.

The second test case is the frigate DTMB in regular head wave either fixed or free to heave and pitch (experimental data came from IIHR). This case is of main interest because there are various experimental data available, including flow measurements. Also measurements at different speeds and wave elevations for similar wave periods are available which allow validating the mesh and solvers for linear extrapolation. Unfortunately this case does not contain a lot of data concerning resistance; to overcome this issue the next two cases have been treated.

The third and fourth test cases are respectively KCS for which experimental study has been conducted by FORCE Technology, and KVLCC2 in regular head waves, both free to heave and pitch. These cases are amongst the most recent test cases and are accurate. For these two cases only, motions and forces have been post-processed.

Table 1: Non dimensionnal sea states used in the study

	Cylinder	DTMB	KCS	KVLCC2
λ/Lpp [-]	8.09	0.50 to 1.50	0.65 to 2.75	0.50 to 2.00
$A_k=\pi H/\lambda$ [%]	14.7 and 21.5	2.5 to 7.5	4.7 to 5.2	1.6 to 6.3

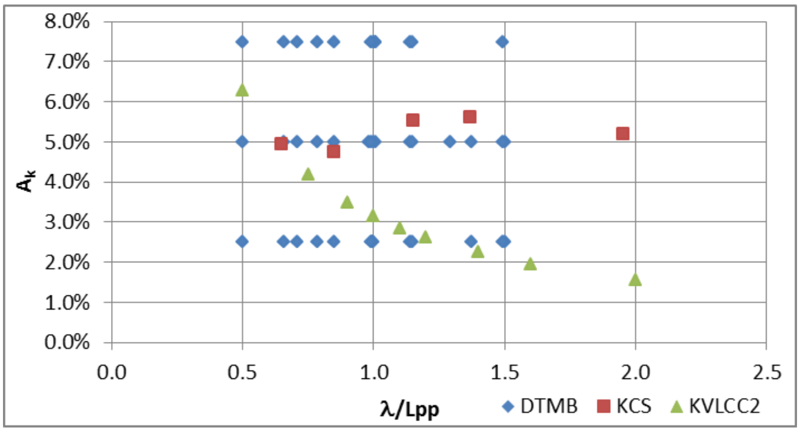


Figure 1: Non dimensional sea states distribution

Table 2: Body mains caracteristics used in simulations

		Cylinder	DTMB	KCS	KVLCC2
Scale		1:1	1:46.588	1:37.890	1:100
Froude number	Fr [-]	0	0.28 0.41	0.26	0.142
Length between perpendicular	Lpp [m]	0.625	3.048	6.070	3.200
Length at water line	Lwl [m]	-	3.052	6.136	3.255
Beam	Bwl [m]	-	0.409	0.850	0.580
Draft	T [m]	0.938	0.132	0.285	0.208
Volume	∇ [m^3]	0.288	0.083	0.957	0.313
Wetted Surface	Sw [m^2]	2.148	1.371	6.618	2.719
Longitudinal position of CoG From aft PP	LCG [m]	-	1.539	2.855	1.710
Vertical position of CoG From keel	VCG [m]	-	0.132	0.378	0.109

3 RESULTS

The results presented in this section are the final submitted results of the three members of the project. The two first cases have been treated by each member, while the two last are treated by only one different participant for each case.

For each test case, the post processing of the different signal (either force or motion) is done by fitting, with a least square method, a sum of harmonically related sinusoids (1) over the three last periods of the signal. For the wave elevation only, the fitting is done against the analytical formulation of a 2nd order Stokes wave in infinite depth (2).

$$h(t) = \sum_{n=0}^{3} a_n \cos(2\pi n t + \phi_n) \tag{1}$$

$$\eta(x,t) = \frac{H}{2}\left(\cos\left(2\pi\left(\frac{x}{\lambda}-\frac{\mathrm{t}}{\mathrm{T}}\right)\right) + \frac{1}{2}\frac{\pi H}{\lambda}\cos\left(4\pi\left(\frac{x}{\lambda}-\frac{\mathrm{t}}{\mathrm{T}}\right)\right)\right) \tag{2}$$

During the realization of the cylinder test case the meshes, time step, and damping zone are investigated in order to obtain minimum error between the simulated wave elevation and the analytical formulation of a Stokes wave, either without the cylinder or with it by taking care of checking waves in an area without diffracted waves.

The cell sizes are varied from 30 to 150 cells per wave length in the direction of propagation and from 5 to 30 cells per wave height in vertical direction. For the lateral direction the cell size should not be more than twice the size in the propagation direction. At last the aspect ratio of the cells (dx/dz) appears to be more important than the actual discretization of the mesh. That is to say, if aspect ratio is too large then the error between the wave elevation and the analytical formulation will be too large even if the longitudinal and vertical discretizations are both correct. This limitation is not the same for each solver.

During this mesh sensitivity analysis, the time step is varied from 50 to 300 time steps per wave period. As expected it appears that the accuracy of the computation is directly linked to the Courant number in the vicinity of the free surface. As for the cell size the limit is not equal for each solver. In STAR-CCM+ the HRIC formulation is used for the convective part of the free surface transport equation. With this scheme the limit due to the blending formulation on the Courant number is set to 0.7. In ISIS-CFD, the BRICS scheme is used. For this one the limit is of 0.3. Both this limitation and a sufficiently large number of time steps per wave period have to be fulfilled in order to correctly propagate the wave.

At last, results for the cylinder are presented in (Table 3, Table 4 and Figure 2). For the two wave heights, the first harmonic of the force signal is similar between simulations and experiments the error is below 1% for 6 simulations out of 9. Besides, the results for the second and third harmonics seem not well predicted. This large error is mostly due to the low values which are compared.

Most of the settings used for the cylinder test case are used for the next three test cases. Time steps are reduced because of the ship motion: the time step discretization is based on the encountered frequency instead of using a discretization per wave period.

For the DTMB 5512 test cases in fixed position, the free-surface and the velocity field at the propeller location are exported each quarter of wave periods. Results are presented on figures 3 and 4. Both the free-surface elevation and the wave patterns are comparable between simulations and experiments. Also the solvers provide really similar results even with different meshes.

The motions simulated for the case DTMB 5512 free to heave and pitch are quite similar to the experimental results. For most of the wave lengths on Figure 5 and Figure 6 there are three simulations and experiments, one for each wave height. These graphs show similar results between CFD and experiments either for the amplitude of the motions and for the phase of the signal.

Finally, on Figure 7 and Figure 8, the transfer functions of motions and added resistance (3) are represented, for respectively the KCS and the KVLCC2 in head waves. In these two cases the three transfer functions fit well between experiments and simulations for the wave lengths close and above the wave length of the maximum added resistance. For smaller wave lengths the added resistance is less similar while motions are still well captured.

Most of these results show good agreement between any CFD solvers and experiments. This is especially satisfying as CFD and experiments match well for various data such as wave elevation, velocity, motions and forces. Also the three solvers provide quite similar data which increases confidence in these tools for solving problems related to waves.

$$ARTF = \frac{F_{wave} - F_{calm,water}}{\rho g (H/2)^2 B^2 / Lpp} \tag{3}$$

4 CONCLUSIONS AND PERSPECTIVES

The study presented in this paper consists of comparisons between the predictions of regular wave interaction with ships, of three CFD codes (STAR-CCM+, ISIS-CFD, and NavalFOAM) against experimental data. The comparisons are done on four test cases: one fixed cylinder and three ships (DTMB, KCS, and KVLCC2) free to heave and pitch in head waves. The numerous experimental data allows setting and validating the methodologies on different aspect of sea keeping, such as wave elevation, motion of ship in waves, and added resistance.

The CFD results presented in this study are mostly in good agreement with experimental data. There are further works to do on small wave lengths modelling for both KCS and KVLCC2 test cases. For these configurations the mesh density has to be increased dramatically while time steps have to be reduced, which leads to an increase of computations cost, and therefore it has not been possible to correctly model these configurations yet.

This validation study was the first step toward a wider use of CFD solvers in the sea keeping field. This work and the following are intended to use CFD in order to answer several questions such as which is the added resistance of ship in a particular sea state, which hull is the most efficient, what are the loads on the hull and superstructures.

There are numerous possibilities to pursue this work. Amongst them will be the investigation of the effects of appendages, and the scale effects which are of great interest to increase our confidence in full scale computations. Moreover the response of ship in irregular sea states has to be investigated. For this purpose the use of SWENSE solvers seems to be a very promising solution. Another work will be developing CFD methodologies for extreme sea states configurations.

REFERENCES

[1] B. Alessandrini , P. Ferrant, L. Gentaz and R. Luquet "Numerical simulation of the 3D viscous flow around a vertical cylinder in non-linear waves using an explicit incident wave model", *Proceedings of OMAE*, (2004).

[2] J. Longo, M. Irvine, F. Stern "Phase-Averaged PIV for the Nominal Wake of a Surface Ship in Regular Head Waves", Asme J. Fluids Eng Vol. 129, pp. 524-540, (2006)

[3] M. Irvine, J. Longo, F. Stern "Pitch and Heave Tests and Uncertainty Assessment for a Surface Combatant in Regular Head Waves", *IIHR – Hydroscience & Engineering*, (2008)

[4] L. Jaehoon, P. Dong-Min, K. Yonghwan "Comparison of Added Resistance for Different Bow Shapes of KVLCC2 Ship", *Ship & Ocean Engineering Research Exchange Meeting*, (2013)

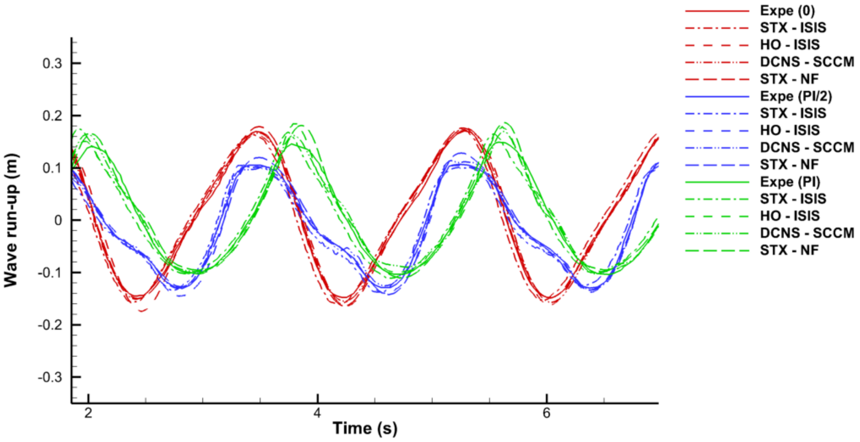


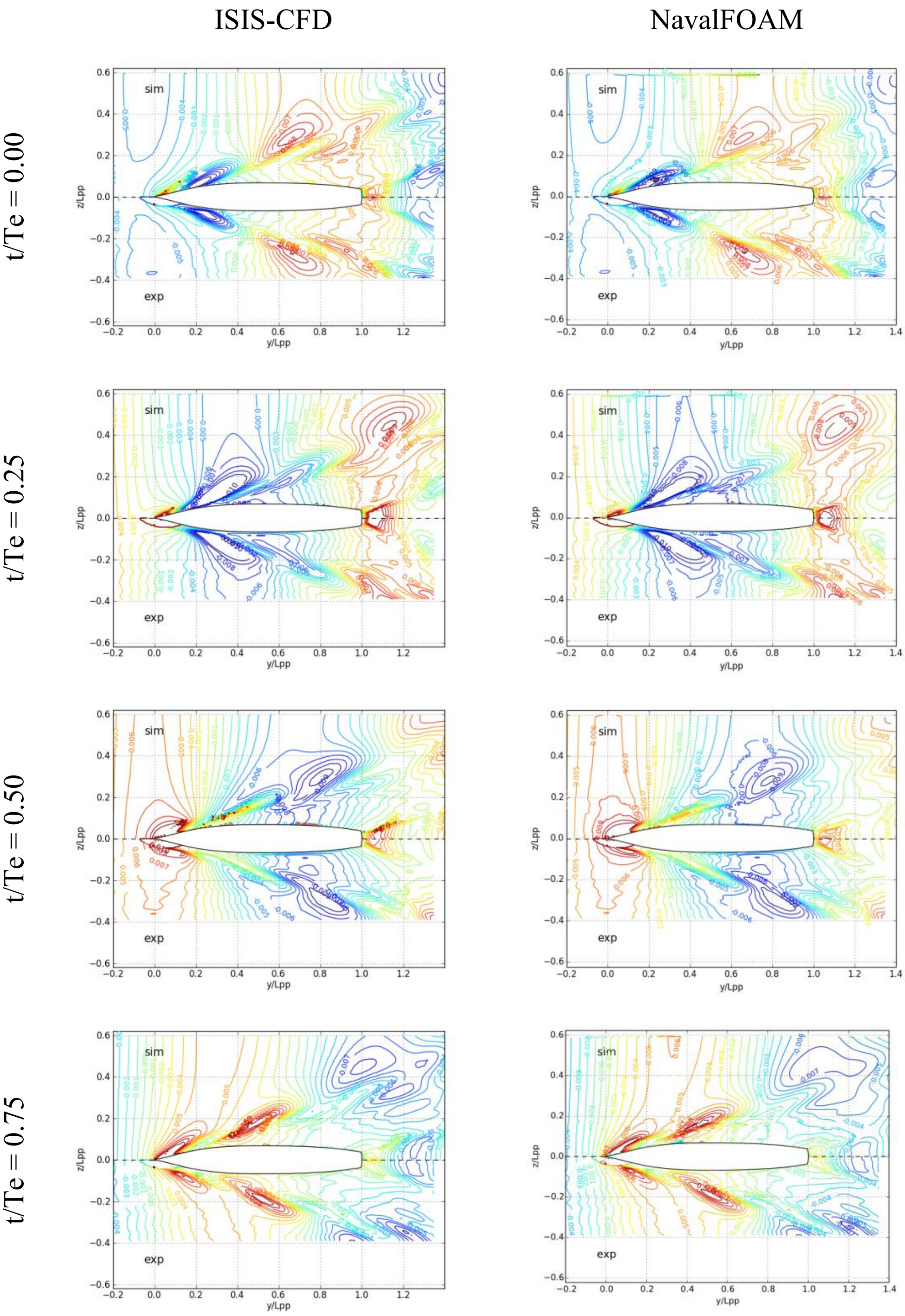
Figure 2: Wave elevation around the cylinder – T=1.8s – H=0.237m

Table 3: Forces on the cylinder – T=1.8s – H=0.237m

	EFD	DCNS STAR-CCM+	HO STAR-CCM+	HO ISIS	STX ISIS	STX NavalFOAM
F1	487 N	-0.2%	-4.5%	0.9%	-0.3%	0.3%
F2	19.8 N	-12.5%	-19.1%	-59.7%	-27.1%	-29.4%
F3	8.2 N	30.4%	14.9%	47.9%	42.6%	36.8%

Table 4: Forces on the cylinder – T=1.8s – H=0.346m

	EFD	DCNS STAR-CCM+	HO STAR-CCM+	HO ISIS	STX NavalFOAM
F1	709 N	-3.1%	-6.0%	-0.1%	0.1%
F2	24.3 N	-0.1%	1.8%	-13.4%	-89.6%
F3	24.2 N	14.2%	15.2%	23.2%	16.4%


ISIS-CFD

NavalFOAM

t/Te = 0.00

t/Te = 0.25

t/Te = 0.50

t/Te = 0.75

Figure 3: Comparison of wave elevation - DTMB

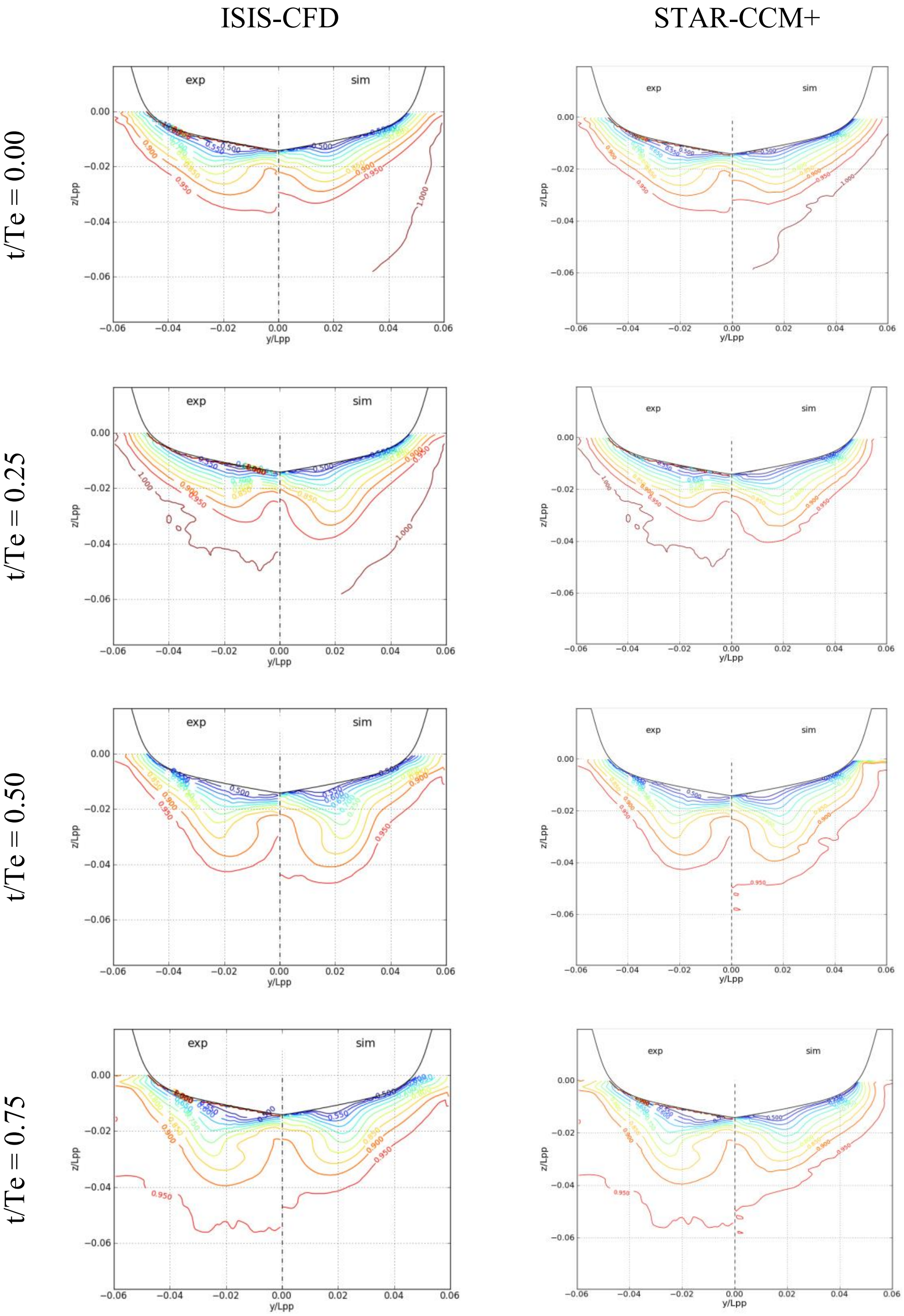


Figure 4: Comparison of velocity level – DTMB

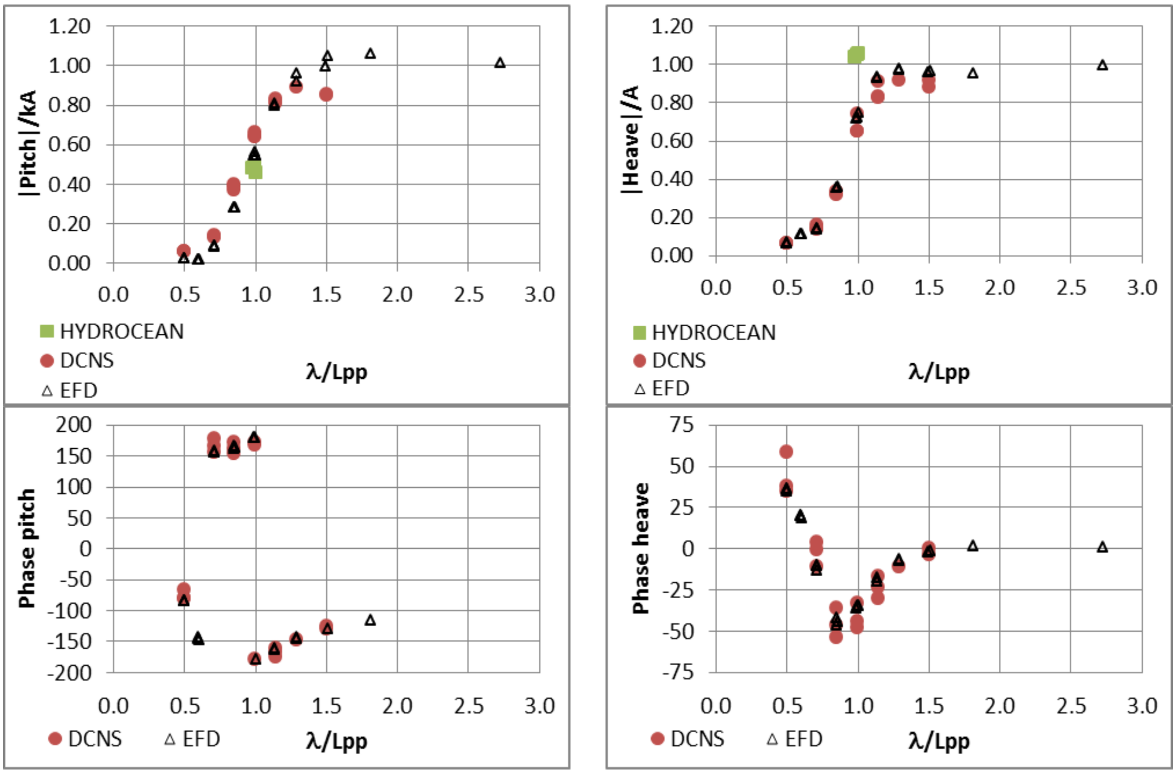


Figure 5: Comparison of heave and pitch in wave – DTMB - Fr=0.28

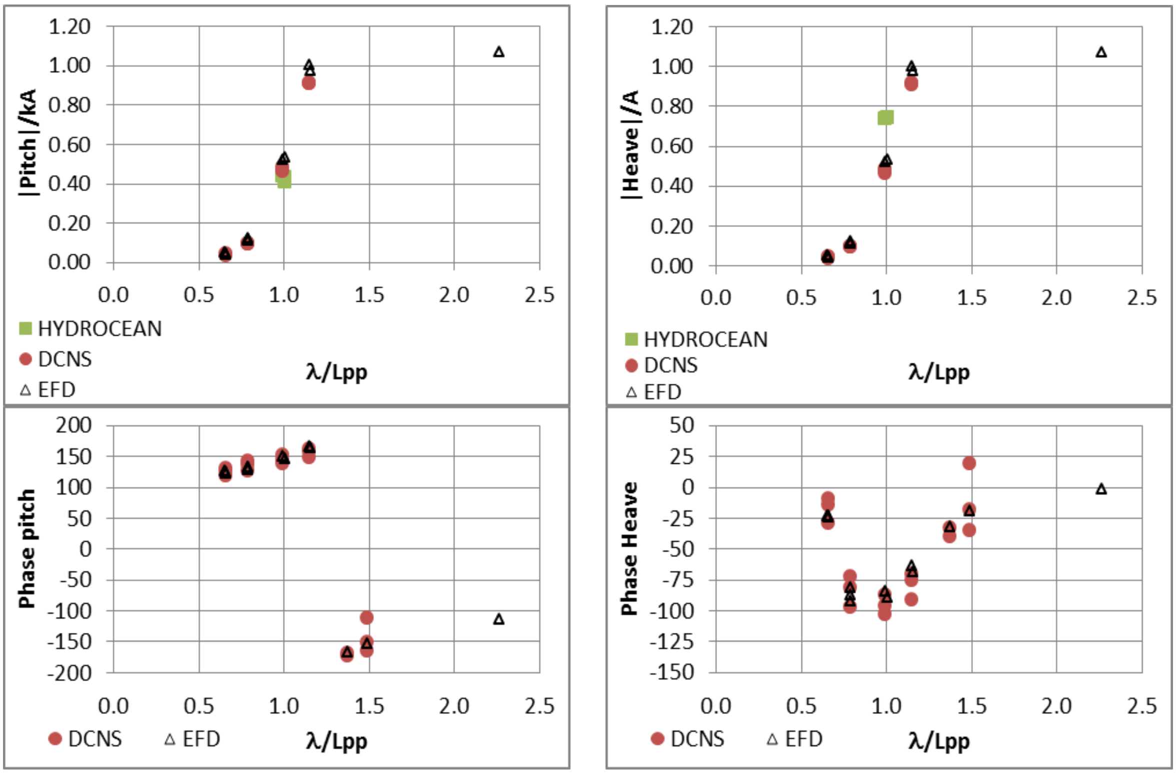
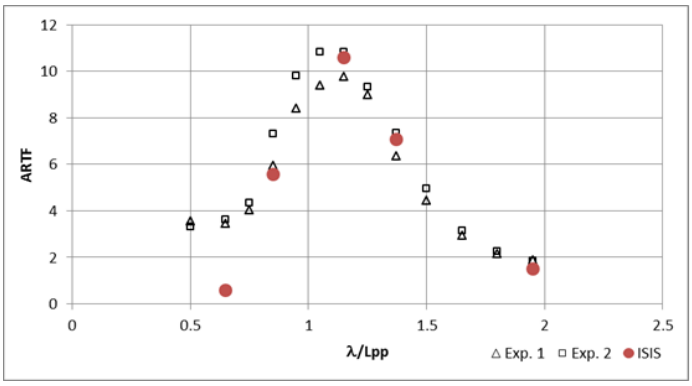
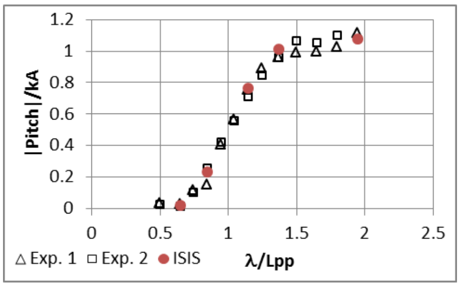


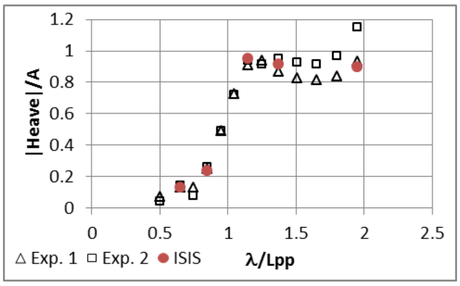
Figure 6: Comparison of heave and pitch in wave– DTMB - Fr=0.41



Added resistance

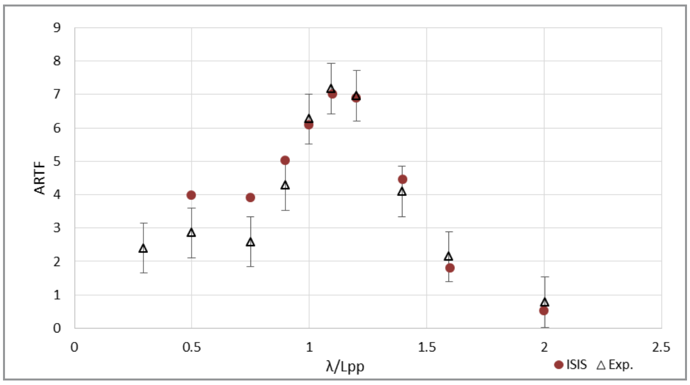


Pitch amplitude

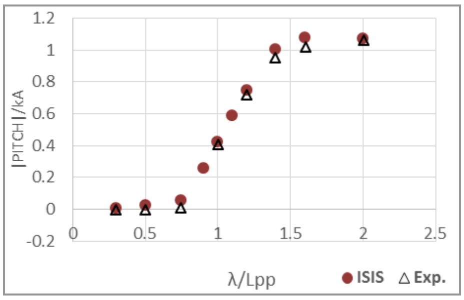


Heave amplitude

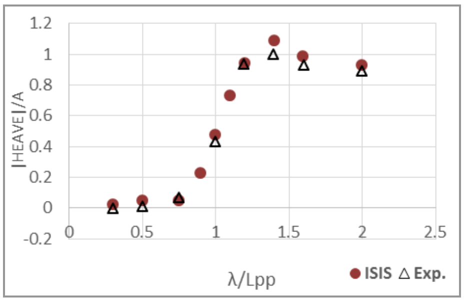
Figure 7: Comparison of added resistance in wave – KCS



Added resistance



Pitch amplitude



Heave amplitude

Figure 8: Comparison of added resistance in wave – KVLCC2