SEA-KEEPING MODELLING IN CALM WATER AND IN WAVES USING THE CFD CODE EOLE

R. MARCER*, C. AUDIFFREN*, C. DASSIBAT*, C. de JOUETTE*, P.E. GUILLERM**

* Principia RD., AREVA Group, La Ciotat
** Bassin d’Essais des Carènes, Val de Reuil

Summary

Sea-keeping numerical simulations are performed with the CFD code “EOLE” developed by Principia and compared with forced oscillations tests in calm water carried out in the B600 basin at Val de Reuil, on a hull model test at 1/24 scale. The numerical code is based on an URANS model and a VOF free surface tracking method. The comparison of CFD results with model tests concludes to a good agreement.

A wave generation model is then included in the CFD code. It consists in splitting the flow field into an incident flow computed by an analytical formulation and a diffracted flow calculated with the URANS model. First validations concerning the problem of the wave diffraction on a vertical cylinder are shown (developments with ECN partnership).

I - INTRODUCTION

The first part of the paper deals with the prediction of viscous roll damping for ship with and without forward speed. In the context of the European VIRTUE program dealing with the improvement of CFD models for naval applications, Principia and Bassin d’Essais des Carènes work on the sea-keeping work-package of the project. It concerns the development and the validation of CFD codes for the simulation of non-linear hydrodynamic effects such as ship roll damping due to viscosity and wave radiation, in calm water or in a wave field.

A key point of the project is the validation of CFD results before using them for hull design. Results exist for simple geometry or for zero-forward speed configurations. However, due to a lack of experimental results, a model test campaign has been required and the most attractive configuration retained was forced-roll motion tests on the Hamburg Test Case hull (HTC). Tests included forced-roll motion for several speeds, several roll periods and magnitudes. Tests were performed on the hull with rudder and bilge keels, hull with bilge keels, and bare hull.

Numerical simulations are issued with the CFD code “EOLE” of Principia. At first, comparisons with forced oscillations tests in calm water are carried out.

The second part concerns the wave generation modeling and validations on the cylinder diffraction problem.

II - THE CFD CODE EOLE

The URANSE software EOLE is a general purpose CFD code developed since 1990 by Principia R&D for incompressible steady or unsteady flows and based on an original pseudo-unsteady system. Developments of EOLE have been directed in particular towards applications relating to steady or unsteady hydrodynamic problems, as for example flow around underwater bodies with non linear free surface, sea-keeping, wave drag, sloshing in tanks.
In order to cope with the numerical estimation of the waves in the discretized fluid domain, the SWENSE technique (ECN) has been implemented in EOLE. It consists in splitting the whole field into an incident flow and a diffracted flow. The incident flow is known by the analytical formulation of Rienecker & Fenton [1] based on the stream function in the case of a regular monochromatic wave (SWENSE Method: Spectral Wave Explicit Navier-Stokes Equations).

The governing equations are the Reynolds averaged Navier-Stokes equations, written in full conservative form consistent with the finite volume integration process on arbitrary-shaped hexaedra.
Following the pseudo-compressibility scheme of EOLE [2], the governing Reynolds averaged Navier-Stokes equations are written :

$$\begin{cases} \dfrac{\partial \tilde{\rho}}{\partial \tau} + div\left(\rho \vec{U}\right) = 0 \\ \dfrac{\partial\left(\tilde{\rho}\vec{U}\right)}{\partial \tau} + \dfrac{\partial\left(\rho\vec{U}\right)}{\partial t} + div\left(\rho\vec{U}\otimes\vec{U} + p\,\overline{I} - \overline{\tau}\right) = \rho\,\vec{f} \\ p = G\left(\tilde{\rho},\vec{U}\right) = \rho\,U_0^{\,2}\ \ln\left(\dfrac{\tilde{\rho}}{\rho_{ref}}\right) + p_{ref} \end{cases}$$

where :
ρ : density of the fluid
$\underline{I}$: the unity tensor
$\overline{\sigma}$: stresses tensor (including the turbulence effects)
$\vec{f}$: external forces
$\tilde{\rho}$: pseudo density of the fluid.
τ : pseudo-time

The introduction of the wave model (SWENSE) is achieved by a decomposition of the physical fields (velocity and pressure) in the following way :

$$\begin{cases} \vec{U} = \overrightarrow{U_p} + \overrightarrow{U_i} \\ p = p_p + p_i \end{cases}$$

where :
$\overrightarrow{U_p}$, p_p are the velocity components and the pressure of the diffracted field
and $\overrightarrow{U_i}$, p_i are the characteristics of the incident waves, which verify the following Euler equations:

$$\begin{cases} div\left(\rho\overrightarrow{U_i}\right) = 0 \\ \dfrac{\partial\left(\rho\overrightarrow{U_i}\right)}{\partial t} + div\left(\rho\overrightarrow{U_i}\otimes\overrightarrow{U_i} + p_i\,\overline{I}\right) = \rho\overrightarrow{f_i} \end{cases}$$

Therefore, the diffracted field is described by the following system :

$$\begin{cases} \dfrac{\partial\tilde{\rho}}{\partial\tau} + div\left(\rho\overrightarrow{U_p}\right) = 0 \\ \dfrac{\partial\left(\tilde{\rho}\overrightarrow{U_p}\right)}{\partial\tau} + \dfrac{\partial\left(\rho\overrightarrow{U_p}\right)}{\partial t} + div\left(\rho\left(\overrightarrow{U_i}+\overrightarrow{U_p}\right)\otimes\overrightarrow{U_p} + p_p\,\overline{I} - \overline{\tau} + \rho\overrightarrow{U_p}\otimes\overrightarrow{U_i}\right) = \vec{0} \\ p_p - p_{ref} = \rho U_0^{\,2}\ln\left(\dfrac{\tilde{\rho}}{\rho_{ref}}\right) \end{cases}$$

The boundary condition on the free surface is written :
$p = p_{atm}$, then $p_p = p_{atm} - p_i$

A *k-ε* model is used to simulate the turbulence of the flow and a coupling with the Gorski model is used for the boundary layer modelling [3].

Two different free surface tracking methods are implemented in EOLE :

- a Volume Of Fluid method based on an eulerian formulation [4]. The time evolution of *F* is governed by a conservation equation : $\dfrac{\partial F}{\partial t} + div\left[\rho\left(\vec{U} - \vec{W}\right)\right]F = 0$, where F is the volume fraction of the denser fluid, $\vec{U} = \vec{U}_p + \vec{U}_i$ and $\vec{W}$ is the grid velocity.
- a mixte eulerian–lagrangian SL-VOF method (Segment-Lagrangian-VOF) developed by Principia [5].

The free surface position can be computed on moving and deforming grids and in a non-galilean reference frame. The VOF algorithm has been extended and adapted on arbitrary curvilinear grids, and two options for the time integration are made available either explicit or implicit. For wave resistance problems, better results are obtained by means of implicit algorithm.

III – EXPERIMENTAL SET-UP

Tests were performed on a model of a container-ship hull-shape. The Hamburg test case model (HTC) comes from HSVA Hamburg. The scale of the model was 1/24 and hydrostatics of HTC are presented on the table 1. Pictures of the model are shown on figure 1.

For tests, bilge keels were added to the model. Those bilge keels were removed for bare hull tests. Moreover to fix the dynamometer on the model an aluminium plate of 1.2 meter length was fix on the model. A specific rudder shaft was manufactured to fix sensors on it. Lastly, pressure gauges were screwed in the hull along a section and next to bilge keel ends.

Displacement volume Moulded (m3)	2.05
Lwl (m)	6.4
Sw (m2)	9.76
Draft (m)	0.43
Width (m)	1.15
Cb	0.652

Table 1 : HTC model hydrostatics

The main objective of the test campaign was to provide experimental data to validate numerical tools on the specific issue of viscous-roll damping. A forced motion is imposed to the model, and forces and torques are measured. In order to prescribe a specific motion to the model, the hexapod was lent by Ecole Centrale de Nantes. Hexapod is a 3D motion generator (figure 2). The hexapod is made of two triangular tray linked by six electric jacks. If one tray is fixed to the ground, for example, then, the other tray can move by operating the jacks.
More details of the experimental set-up can be found in [6].

III - COMPARISONS CFD-EXPERIMENTS FOR SEA-KEEPING APPLICATIONS

The test cases concern the HTC hull in a forced roll motion and with or without a constant forward speed.

III.1 - Modelling assumptions

The simulations are based on the following assumptions :

- Navier-Stokes turbulent (k-ε) model
- VOF implicit free surface model
- Imposed motion :
  - on the ship surface mesh : forced roll oscillations (period T) with or without forward speed.
  - the global mesh moves with the ship in the absolute frame. For the case with forward ship, the velocity is imposed to the body in a fluid at rest.
- Simulation duration : 6*T
- Time step = 1/200 T
- HTC model at scale 1/24.
- Initial conditions : fluid at rest
- Open boundary conditions allowing to damp the radiative waves induced by the ship motion.

The structured multi-blocks mesh of the hull is composed of 11 blocks and about one million of cells. Figure 3 shows the discretized geometry and a zoom of the mesh at the stern.

Figure 1 : view of the HTC

Figure 2 : hexapod – a 3D motion generator

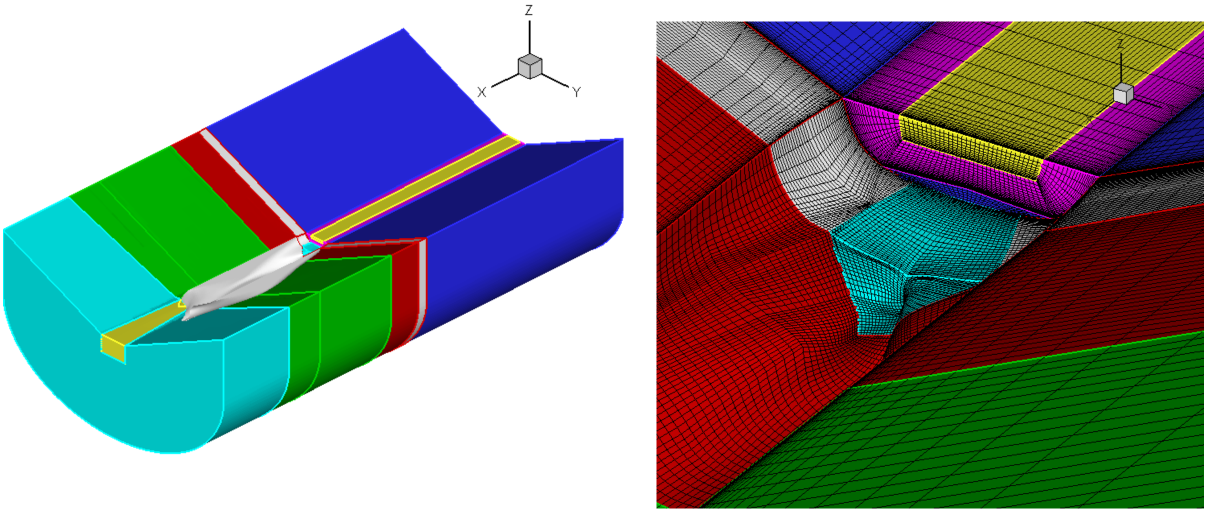

Figure 3 : multi-blocks mesh of the HTC – zoom on the stern

III.2 - Results of the moment loads

Figures 4-6 show comparisons simulation-experiment of the hydrodynamic moments time evolution.

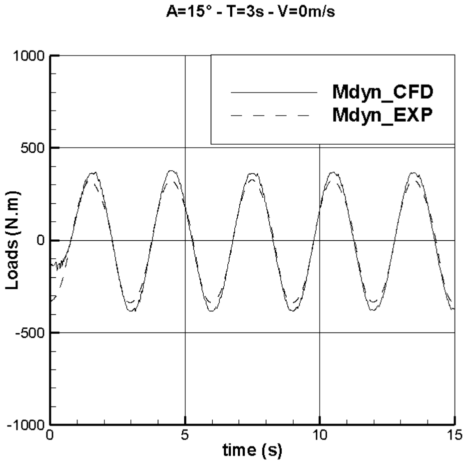

Figure 4 : Roll period = 3s – roll angle = 15° - forward speed = 0 – bare hull

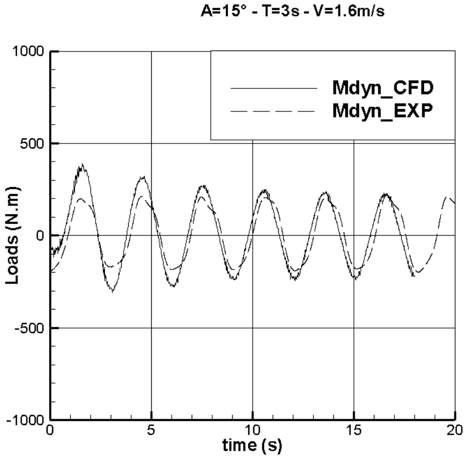

Figure 5 : Roll period = 3s – roll angle = 15° - forward speed = 1.6m/s – hull with bildge keels and rudder

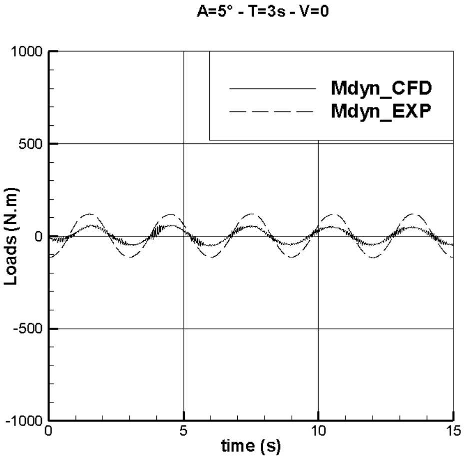

Figure 6 : Roll period = 3s – roll angle = 5° - forward speed = 0 – bare hull

The behaviour of the measured and computed loads is, on the whole, very similar. Indeed, the roll moments remain perfectly periodic in time and the phase remains identical.
Experimental and computed roll moments signals globally have the same order of magnitude.
With the forward speed case (figure 5), maximal discrepancies appear at peak amplitude but remain smaller than 5% for the hydrodynamic moment. The damping effect due to bilge keels seems to be well predicted by the CFD model.
The maximum error on the hydrodynamic moment is obtained with the smaller roll angle (figure 6). However, this case does not lead to significant experimental values because measurements of small loads amplitudes in the case of a bare hull without forward speed are not very consistent.

IV – VALIDATION OF EOLE FOR WAVE MODELLING

IV.1 Monochromatic wave propagation

The propagation of incident waves is studied in an open parallelepiped fluid domain. The purpose of this computation is to reproduce numerically with the VOF method the monochromatic waves propagation deduced from the analytical theory provided by Rienecker and Fenton [1]. The aim is to check the accuracy of the VOF method, regarding to the shape waves and losses of amplitude, according to the analytical solution.

	Characteristics of the test case
Crest to trough height	0.10 m
Wavelength	1.56 m
Period	1 s
Depth	1m
Δt	0.02s
Simulation duration	500Δt=10s=10T

Table 2 : characteristics of the test case

The mesh includes (157*5*71) cells in (X,Y,Z).

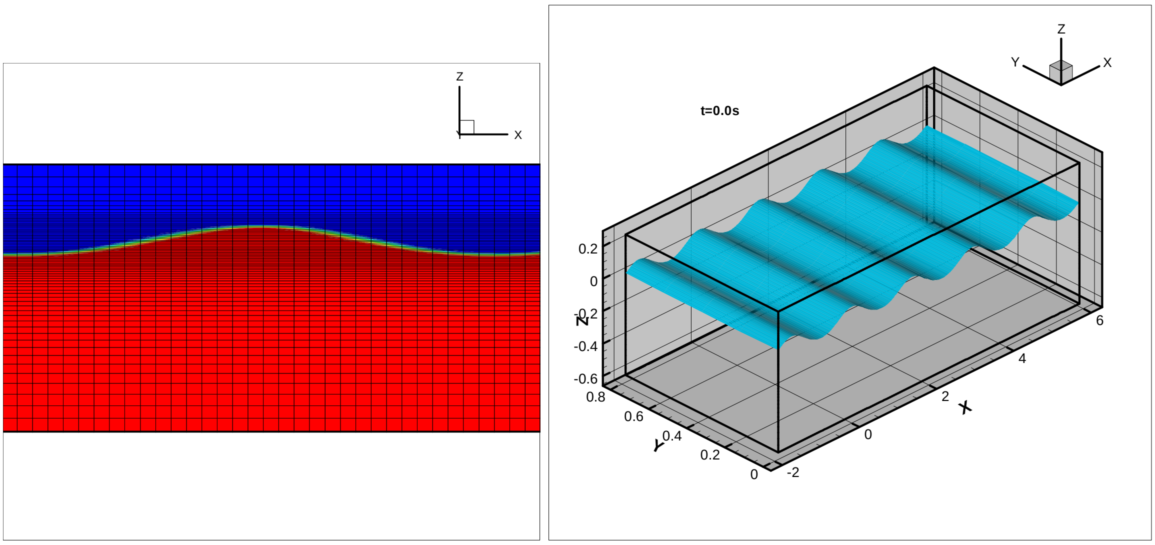

Figure 7 : zoom of the mesh on a longitudinal plane and initial wave condition

Initial conditions for free surface elevations, velocity components and pressure are obtained with the initial solution provided by Rienecker & Fenton.

The boundary conditions concern only the diffracted field :

- lateral boundaries : slip condition
- inlet and outlet boundaries : periodic condition. This condition allows to have reasonable size of the fluid domain. The only constraint is that the domain dimension in the propagation direction must be proportional to the wave length.

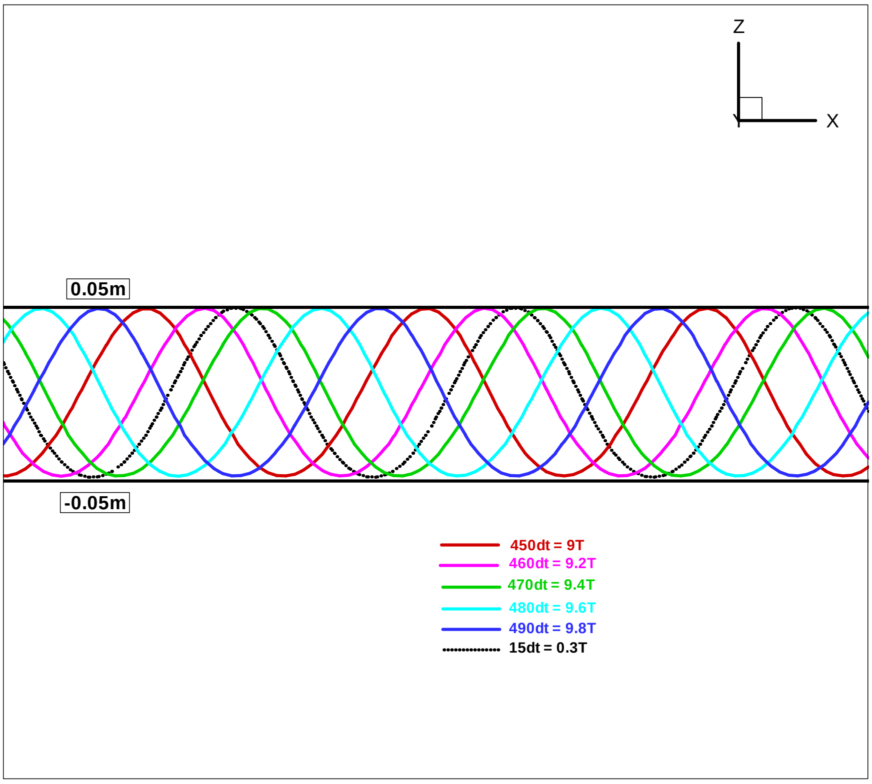

Figure 8 : position of the free surface at the beginning of the simulation (--- 0.3T) and between 9T and 10T

Figure 8 shows that the free surface remains perfectly sinusoidal during the simulation, with an amplitude strictly included between –0.05m and 0.05m (initial condition). There is strictly not any loss of the wave amplitude after 10T (500 Δt). These results validate the coupling of the Navier-Stokes-VOF algorithm of EOLE with the SWENSE method.

IV.2 - Wave diffraction on a cylinder

The cylinder is supposed vertical and fixed on the seabed.

	Characteristics of the test case
Crest to trough height	0.1 m
Wavelength	1.56 m
Period	1 s
Depth	1m
Diameter of the cyl.	0.6m
Δt	0.02s
Simulation duration	300Δt=6s=10T

Table 3 : characteristics of the test case

The mesh includes (50*20*98) cells in (X,θ,Z) (figure 9).

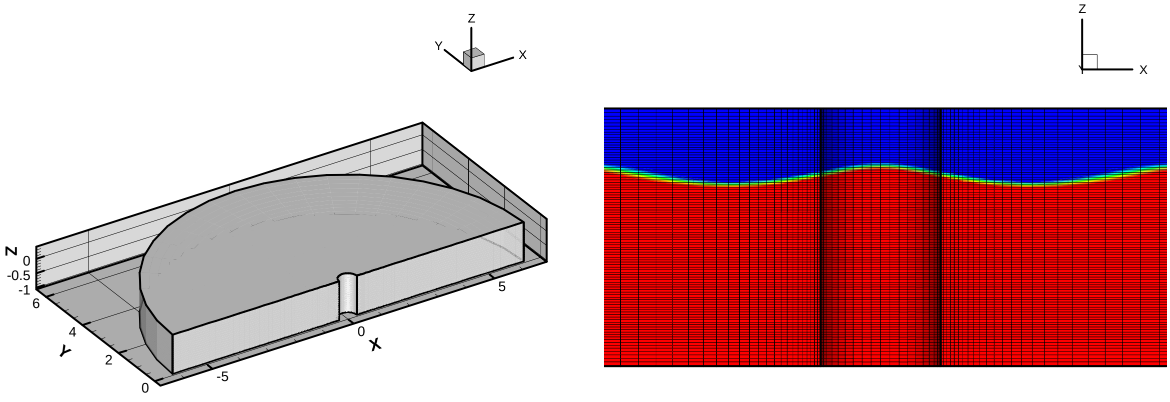

Figure 9 : view of the geometry - zoom on the meridian plane

The boundary conditions concern :

- Symmetry conditions on the meridian plane
- Cylinder : no slip condition for the total velocity
- Outer boundary : diffracted field = 0

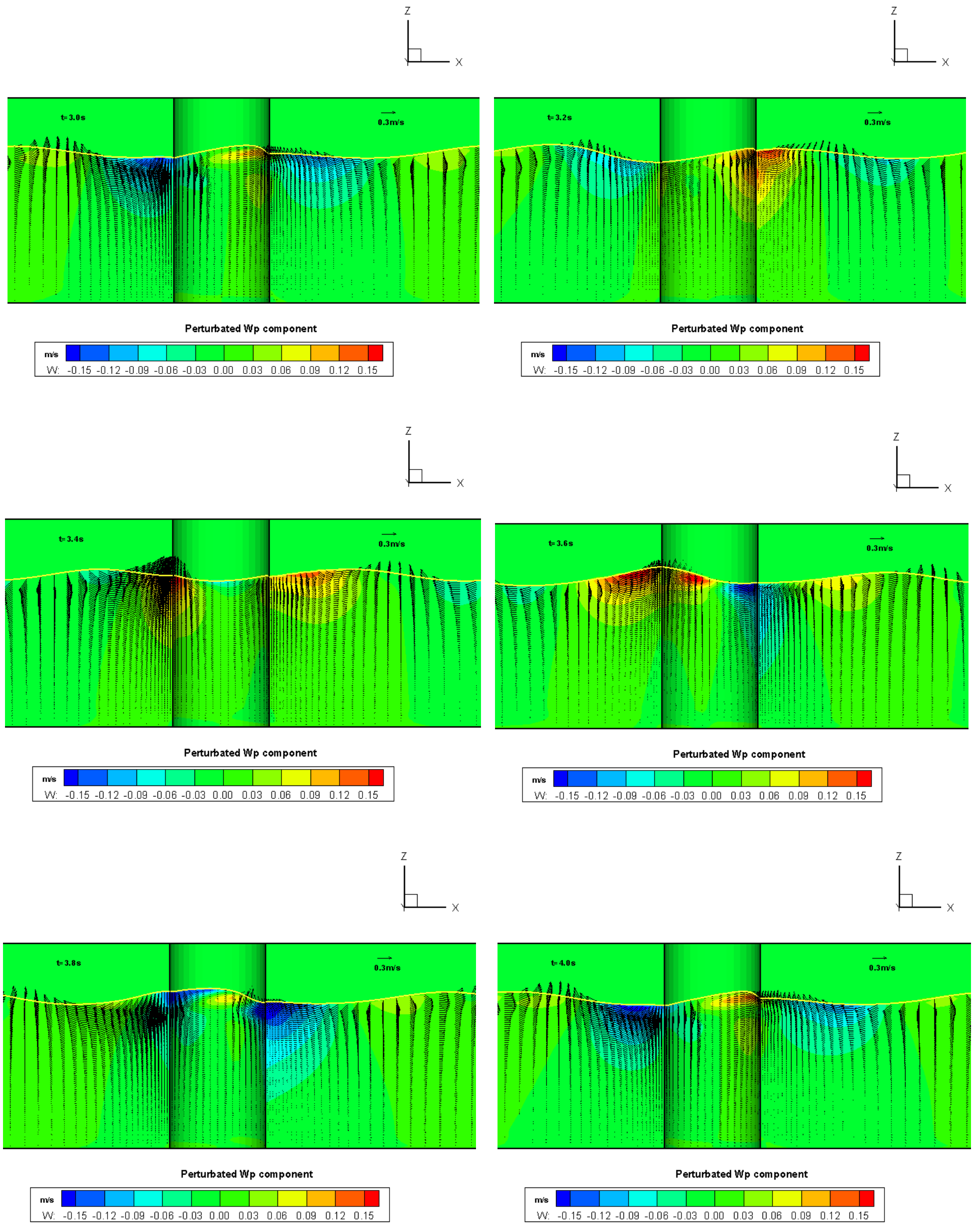

Figure10 : waves evolution and diffracted velocity (W component) during a period

The wave diffraction is concentrated around the cylinder.
Note that the perturbed flow remains periodic even in the wake of the cylinder (same pictures at t=3s and t=3s+T=4s).

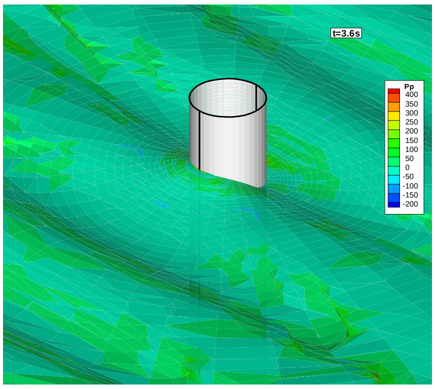

Figure 11 : 3D visualisation of the wave field

Figure 13 shows the time evolution of the drag load issued from EOLE and the comparison with analytical results based on the potential theory [7].
The maximal intensity reached by the load peaks is comparable to the load value given by the potential theory.

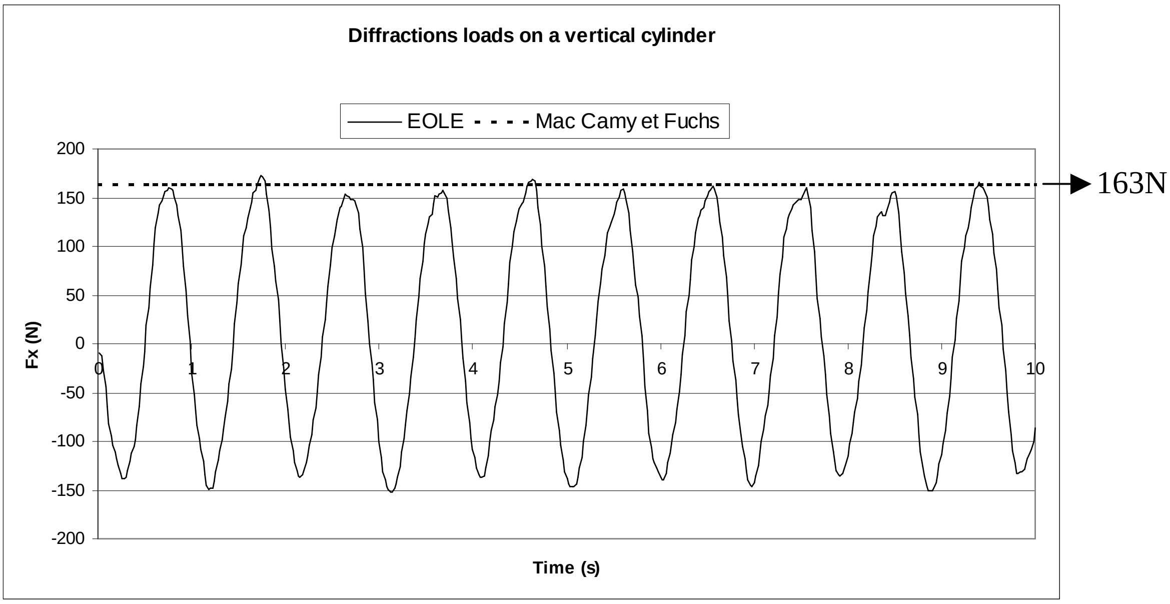

Figure 12 : comparisons simulation-analytical formulation (Mac Camy et Fuchs [7]) of the loads applied on the cylinder

V - CONCLUSIONS

Hydrodynamics validation works of the 3D CFD code EOLE have been carried out.
Firstly, they concerned comparisons with measurements of forced oscillations tests performed on a hull model test at 1/24 scale, and on wave modelling. On the whole, the results are satisfactory especially for the hull with appendages (bilge keels) for which the damping, related on vortices development and wave effects at the free surface, is well reproduced by the code and its turbulent model.

In second, validations for wave modelling applications have been issued. They show the ability of the code to simulate wave diffraction problems by fixed structures.
Next validations will concern diffraction phenomena on moving structures (ships).

ACKNOWLEDGEMENTS

This work was supported by the European VIRTUE project within the 6th Framework Program "sustainable development, global change and ecosystems". The authors thank C. Berhault and P. Guerin for many fruitful discussions, G. Barras for his contribution in mesh works, and B. Pettinotti for his contribution to the model set-up.

REFERENCES

[1] Rienecker M.M., Fenton J.D., A Fourier approximation for steady water waves, *Journal of Fluid Mechanics,* 104:119-137, 1981.

[2] De Jouëtte (C.), Viviand (H.), Wornom (S.), Le Gouez (J.M.)., Pseudo compressibility method for incompressible flow calculation, 4th International Symposium on Comp. Fluid Dyn. Of California at Davis, sept. 12, 1991.

[3] J.J. Gorski, A new near-wall formulation for the k-ε equations of turbulence, *AIAA Journal,* 1986.

[4] Hirt (C.W.), Nichols (B.D.), Volume of Fluid Method for the dynamics of free boundaries, *J. Comp. Phys.* 39, 1981.

[5] Biausser, B., Guignard, S., Marcer R., and Fraunié, P, (2004), 3-D two-phase flows numerical simulations by SL-VOF method, *Int. Jour. For Num. Meth. In Fluids,* Vol 45, pp 581-604.

[6] Marcer, C. Audiffren, C. Dassibat, C. de Jouëtte, P.E. Guillerm, B. Pettinotti, (2007), Validation of a CFD code for sea-keeping simulation, Paper No ISOPE-2007-PF-01.

[7] Mac Camy R.C. and Fuchs R.A.. Wave Forces on Piles : a Diffraction Theory. U.S. Army Corps of Engineers, beach Erosion Board, Tech. Memo No. 69.