First International Symposium on Fishing Vessel Energy Efficiency
E-Fishing, Vigo, Spain, May 2010

HydroPêche: a way to improve energy efficiency of fishing devices

Grégory Germain [1], Philippe Druault [2], Roger Lewandowski [3], Benoit Vincent [4],
Daniel Priour [5], Jean-Yves Billard [6]

1 Hydrodynamic & Metocean Service, IFREMER, 150 Quai Gambetta, 62321 Boulogne-sur-Mer, France.
2 University Pierre et Marie Curie-Paris 6, IJLRA, Case 162, 4 Place Jussieu, 75252 Paris cedex 5, France.
3 IRMAR, Campus Baulieu, Univ. Rennes 1, 35000 Rennes, France.
4 Marine Technology Laboratory IFREMER, 56 100 LORIENT, France
5 Marine Technology Laboratory, IFREMER, BP 70, 29280 Plouzané, France.
6 IRENAV, Ecole Navale, 29240 Brest, France.

Abstract—**The project presented here and named HydroPêche aims to develop a tool for an automatic optimisation of trawl in order to minimize the drag of the gear. For that purpose, different aspects are studied in order: a/ to extend the basis of experimental data on flow characteristics governing the hydrodynamic behaviour of different porous structures ; b/ to develop numerical tools to simulate more realistic flow around porous structures taking into account fluid / structure interactions; c/ to develop automatic optimisation tools to design efficient trawls in terms of energy consumption.**

The developed tool should provide a substantial gain on the fuel consumed of actual fishing devices while maintaining a comparable fishing efficiency. The developments made will also be extended to study different types of porous structures used for fishing and aquaculture devices and to better understand the development of selectivity devices. Initial results have been achieved: some specific data analysis was carried out from PIV measurements around a cod-end, FreeFem ++ has been used to simulate the flow around a specific cod-end and the first developments of an automatic optimisation tool give good numerical results in terms of energy efficiency for both bottom and pelagic trawls.

Keywords – Fluid Mechanic; Experimental trials ; Numerical Simulation; Bottom Trawl; Optimization; Drag Reduction

I. INTRODUCTION

Trawls, being one of the most common fishing gears, are subjected to numerous studies devoted to energy efficiency improvement. Considering that the drag is a function of the towing speed many fishermen reduce this parameter in order to lower fuel consumption. Recently, new twine materials have been tested in some parts of the trawls with the aim of reducing twine diameter and therefore the drag. Ward & al. [1] studied trawls involving novel materials, which led to a drag cut down of 6% and a mouth opening increased by 10%. Parente & al. [2] have improved bottom trawls by using larger meshes and by changing the panel cuttings, which generated fuel reductions of up to 18% and a potential increase of the net cash flow up to 27%.

The efforts (time and money) to carry out such works are very important and the need for numerical tools for trawl optimisation is of great interest to increase energy efficiency. Indeed, the hydrodynamic behaviour of porous structures used for bottom or midwater trawls is still poorly understood. The complexity of the flow makes numerical simulations difficult and today no effective model exists to simulate trawl behaviour during fishing operations.

The goal of the work carried out under the HydroPêche* project is to develop a tool for automatic optimisation of trawls in order to minimize the drag of the gear. For that purpose, different aspects both experimental and numerical are studied in order:

- to extend the basis of experimental data on flow characteristics governing the hydrodynamic behaviour of different porous structures (sheets of net, trawls)
- to develop numerical tools to simulate more a realistic flow around porous structures taking into account fluid / structure interactions;
- to develop automatic optimisation tools to design efficient trawls in terms of energy consumption (value of the drag of the gear by respecting a number of both economic and environmental constraints) and to adapt the numerical code for the routine used by an optimiser.

---

* Project supported by the European Fisheries Fund and the French Ministry of Agriculture and Fisheries

The developed tool should provide a gain of few 10% on the fuel consumed by actual fishing devices while maintaining a comparable fishing efficiency. The developments made will also be extended to study different types of porous structures used for fishing and aquaculture devices (nets, cages, seines) and to better understand the development of selectivity devices.

This project is conducted by Ifremer (French Research Institute for Exploitation of the Sea) in partnership with: Ecole Navale, Ecole Centrale de Nantes, University of Paris 6 and University of Rennes 1, and to date four PhD students have worked on the different tasks. Since the beginning of the project in 2009, the initial results have been achieved: some specific data analysis were carried out from PIV measurements around a cod-end, the numerical tool FreeFem++ [8], [9] has been used to simulate the flow around a specific cod-end and the first developments of an automatic optimisation tool give good numerical results in terms of energy efficiency for both bottom and pelagic trawls.

In this paper we present a summary of these works in three different parts: a/ experimental flow characterisation around trawl and net structures, b/ numerical simulations for trawl behaviour modelisation and c/ energy optimisation of bottom and pelagic trawls.

II. EXPERIMENTAL FLOW CHARACTERISATION

This section is devoted to the first aspect of this project, *i.e.* the experimental investigation of the turbulent flow around bottom trawls. With a better knowledge of the flow characteristics, the improvement of the structural analysis of a net structure taking into account the effect of the trawl on the inflow condition will be possible. As well known, the flow and thus the drag forces on net structure are strongly dependent on both the geometry (Solidity ratio) and the Reynolds number [3]. If it is evident that for increasing Solidity ratios, flow interaction will become increasingly significant, these disturbances are not well known and easily characterized. So, in order to evaluate these effects and to identify the areas where the drag is generated, we have carried out an experimental work to characterise the flow characteristics around two kinds of structures: a/ a 1/10 scale bottom trawl and b/ a rigid cod-end (Figure 1). The bottom trawl is used in order to determine the evolution of the boundary layer along the structure while the rigid cod-end is used in order to characterise the turbulent flow in this specific region.

The flow is characterised from Particle Image Velocimetry (PIV) measurements and the trials carried out in the Ifremer wave and current flume tank (Figure 2). Another paper [4] sets out precisely both the experimental results and the tools used for the post processing (Proper Orthogonal Decomposition - POD) and the analysis (spectral analysis) of the experimental data. We only give here the principal results.

The control of boundary layers or more generally the hydrodynamics instabilities of the near wake past the porous structure in high Reynolds number flows, is still a matter of central interest in our HydroPêche project devoted to the reduction of drag of the gear. Indeed, the motivation of current study is to achieve drag reduction but also to provide additional information about the turbulent flow characteristics of the flow around the trawl.

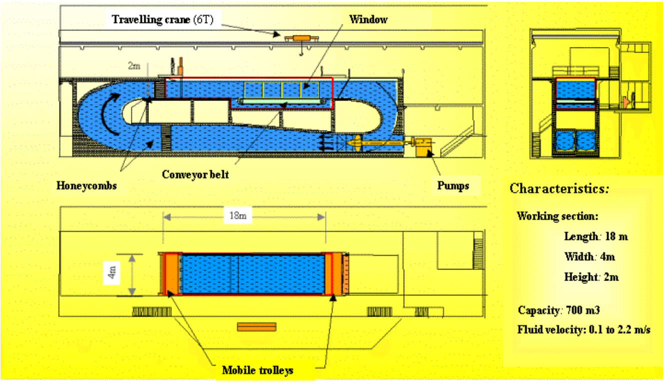

Figure 1. Ifremer free surface hydrodynamic water tunnel.

The characterisation of the boundary layer evolution along the net has been attempted around a 1/10 scale bottom trawl. Unfortunately, the movements of the trawl have made PIV measurements non exploitable. Some tests with a lower turbulent intensity of the incoming flow for the restriction of the movement of the trawl will be attempted. The characterization of the evolution of the boundary layer along the trawl may be done using these new measurements.

From these trials, only the analysis of the motion of the cod-end, as shown in Figure 2, demonstrates the need to study specifically this part of the structure in order to identify how this high turbulence flow area can be a source of drag generation.

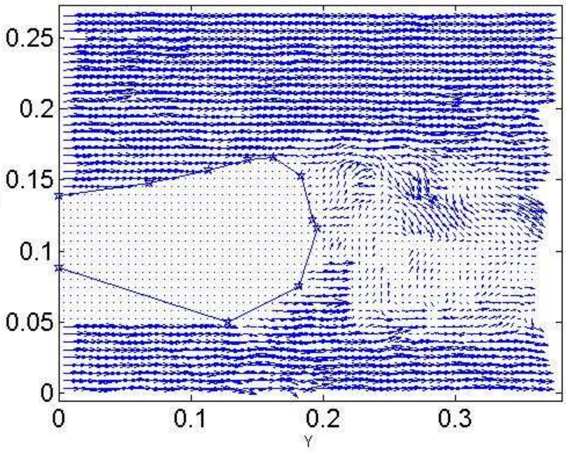

Figure 2. Instantaneous fields around the bottom trawl cod end.

In order to achieve this aim, a specific rigid cod-end was built to avoid instabilities during flume tank tests [5]. This structure is perfectly stable in the flow and its geometrical shape is known (Figure 3). With the use of this structure, the main characteristics of the flow around this specific part of a trawl can be studied. The global characterization of the flow including the spatial representation of the mean flow field

(Figure 4) and the mean kinetic energy (Figure 5) may be quite interesting for the numerical computation of the drag coefficient of such a porous structure.

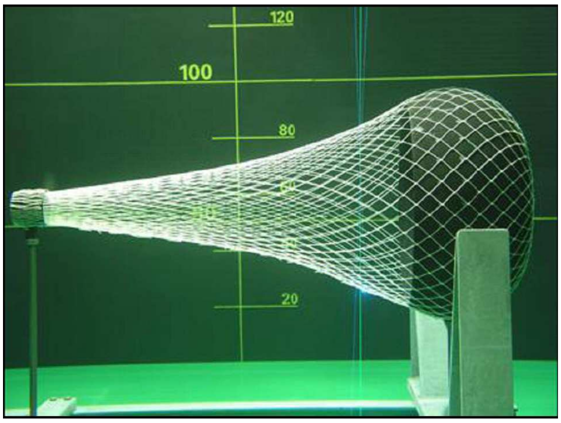

Figure 3. Rigid cod end in the flume tank

In Figure 4, the vertical patterns of the wake are clearly visible. Indeed, the near wake of the porous structure is dominated by two large counter-rotating vortices which are symmetric along the main axis of the structure. To access the turbulence levels of the flow, the mean turbulent kinetic energy is shown in Figure 5. As it has been observed in Figure 4, the turbulent wake is clearly identified. Indeed, the separated shear layer is observed in this figure.

In this work, to investigate the large scale flow structures of the wake of the porous structure, the Proper Orthogonal Decomposition (POD) is used. POD was first applied in fluid mechanics in 1967 by Lumley [6] in order to extract in an objective way the large scale flow structures present in a turbulent flow. This procedure is usually based on the computation of the eigenfunctions of the two point correlation tensor of the velocity field. POD eigenfunctions are arranged according to their energy content and the first POD modes are associated with the large scale structures of the flow. More details about this procedure and its application can be found in [7].

The implementation of the POD procedure performed on from the whole set of available PIV instantaneous velocity vectors fields has proven its effectiveness in showing the main flow organization of such a turbulent flow by the extraction of the energetic large scale coherent structures present in the wake of the structure. The first mode and the first 44 ones contain respectively 25 % and 80 % of the total kinetic energy. An illustration of the potential of this methodology is given on Figure 6, where the projection onto the 44 POD modes corresponding to 80% of the total energy of a raw instantaneous velocity vector field is given. Quasi-similar results can be obtained for each instantaneous velocity field projected onto these first POD modes. So, from these results, it is possible to extract the coherent structure embedded in the background turbulent flow and thus to dynamically follow these structures and to estimate their dynamical space and time evolution (see [4] for more details).

Furthermore a spectral analysis allows the investigation of the vortex shedding frequency of this flow. Such analysis associated with a porous structure has not yet been fully investigated. The particularity of trawl or related fishing nets concerns the difficulty in numerically modeling these porous structures properly. Nowadays such an investigation can be carried out essentially from experimental measurements. From our results, the turbulent wake flow behind the cod-end can be related to previous analysis of a turbulent wake flow behind a sphere and/or a cylinder [10], [11].

Even if these results give some very interesting new information about the turbulent flow behind a cod-end, an extension of this study has to be performed in order to characterise the turbulent flow around the overall structure of both bottom and pelagic trawls. Measurements around a bottom trawl are under investigation and similar analyses to those presented in this paper will be carried out.

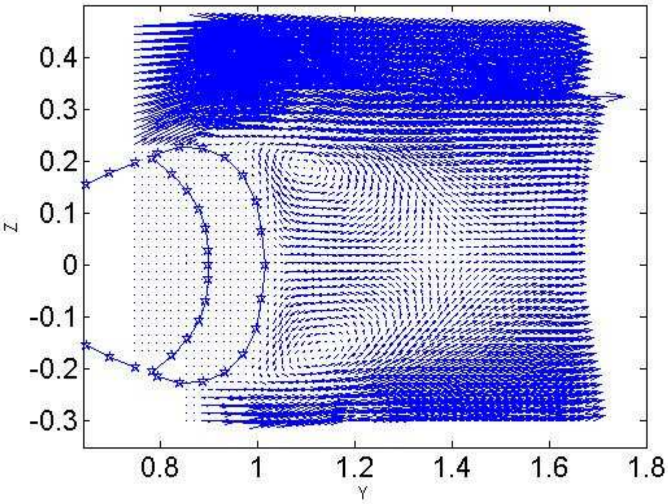

Figure 4. Mean velocity field behind a rigid cod-end.

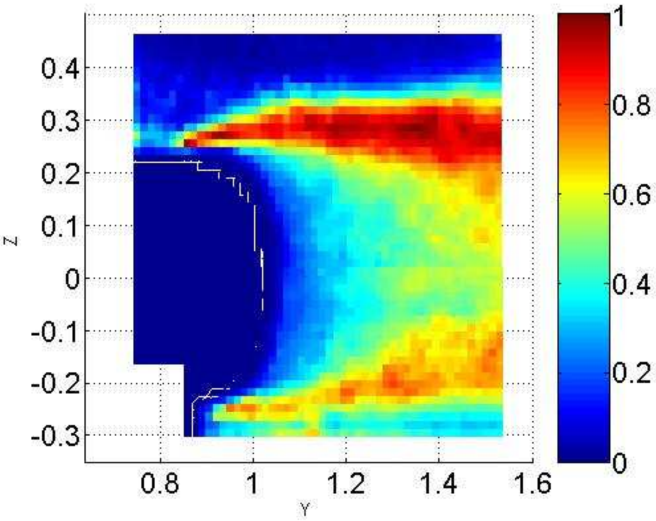

Figure 5. Normalized turbulent kinetic energy behind a rigid cod-end.

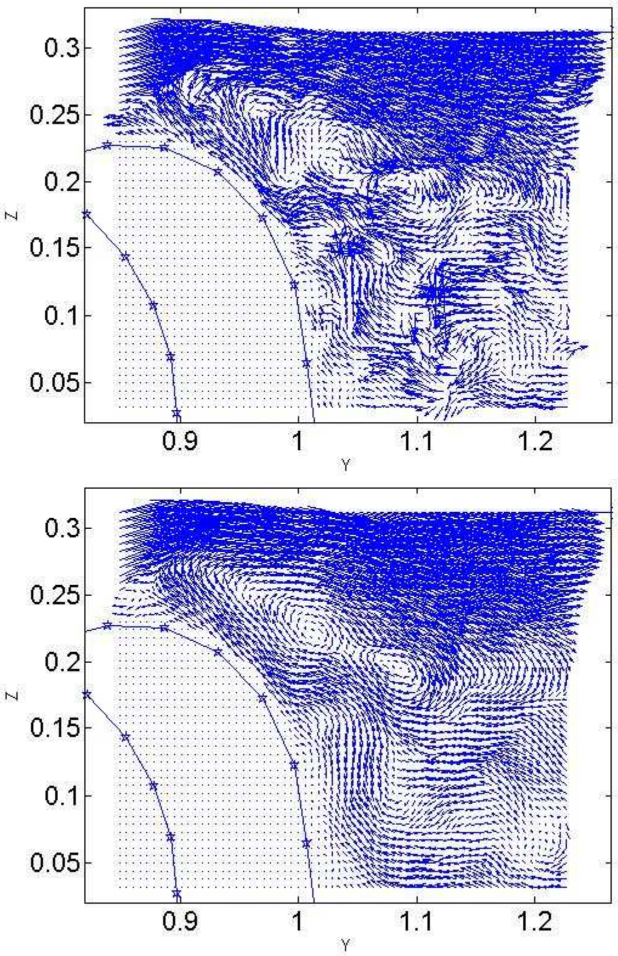

Figure 6. Top: Instantaneous PIV velocity field . Bottom same instantaneous PIV velocity field projected on the first 44 pod modes.

The POD method is here employed to extract dominant coherent structure from experimental data, but this method can also be used in a reconstruction process to obtain flow entry conditions for numerical models.

III. NUMERICAL SIMULATIONS

The mechanical system made of the elastic net alone in a given laminar uniform flow with very simple interaction laws has been studied, see for instance [8] and [9]. For the majority of them, the influence of the flow is only taken into account by the use of Landweber / Richtmeyer laws to calculate the drag force exerted on each twine of the net panel constituting the trawl:

$$F = \frac{1}{2}\rho CdDL(V\sin\theta)^2$$

$$T = f\,\frac{1}{2}\rho CdDL(V\cos\theta)^2$$

with F the normal force to the twine, T the tangential one, ρ the mass density of water, Cd the normal drag coefficient, f the tangential coefficient, D the diameter of the twine, L the length of the twine, V the upstream velocity, θ the angle between the twine and the current.

These mechanical models do not take into account any fluid/structure interaction, *i.e.* the flow is not disturbed by the net structure. First simulations approach of the flow around an axisymmetric rigid net has already been carried out in [3] and [12] but they can not be extensible to the 3D case of an elastic net in interaction with a flow. So, today no numerical simulation of the complete system net/flow, with a real interaction coupling exists.

Knowing that the local speed of the flow has a direct impact on the trawl shape and then the mesh opening, it is essential to develop flow modelling. Due to the complexity of this task, the cod-end case is chosen for the development of our numerical model. The experimental work presented above shows that the flow to simulate is turbulent. Direct Numerical Simulation would not be able to treat a problem with such a high Reynolds number (here ~105, using as reference length the maximal diameter of the catch, i.e. 0.45 m, and the entrance velocity as reference velocity that is equal to 0.51 m/s). So, a model based on a Reynolds Averaged Navier–Stokes turbulence model penalized by a term based on the Brinkman law has been initiated by [13], [14]. The extension of the fully 3D case is currently under progress. Figure 7 show the 3D numerical domain used for these developments. The net is considered as a fictitious domain (Figure 8) and the fluid equations are solved in the flow domain as well as the net domain. With a right choice of the permeability function the model yields numerical simulations which fit very well with the experimental data: Figure 9 shows the comparison of the vertical velocity profiles obtained numerically and experimentally, while the distribution of the turbulent kinetic energy is given in Figure 10.

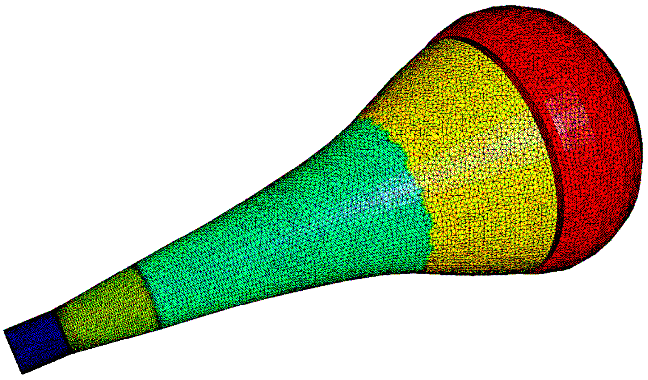

Figure 7. Description of the 3D numerical domain.

These first simulations have been carried out using the free software FreeFem++ [15]. It allows computations of 2D and axisymetric fluid dynamics by the means of the finite elements method. This software is currently under progress and the first 3D simulations to obtain the flow around a rigid cod-end are today considered. Once these simulations are validated, a coupling between the mechanical model and the fluid one should be envisaged in order to simulate the complete fluid/structure interaction system.

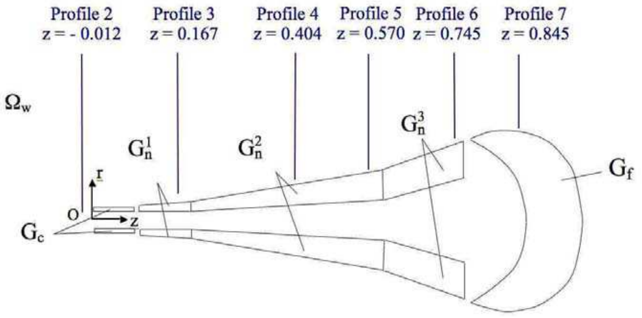

Figure 8. Description of the numerical domain and LDV profiles for experimental validation.

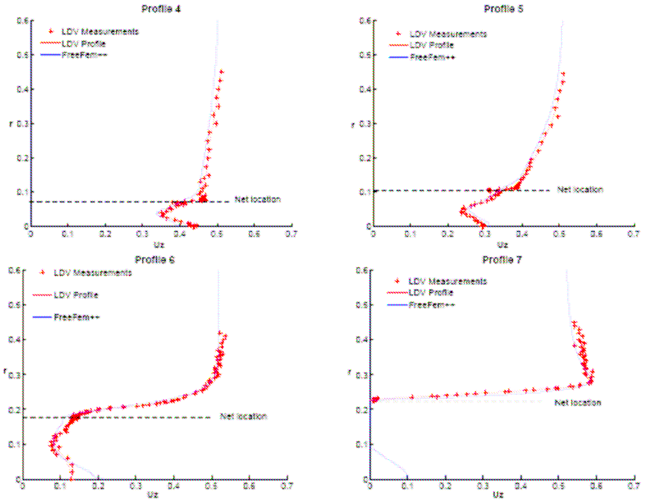

Figure 9. Comparisons between numerical and experimental results.

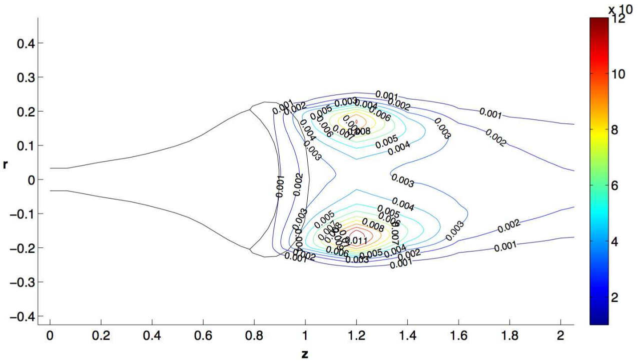

Figure 10. Repartition of the turbulent kinetic energy.

IV. OPTIMISATION WORK

Despite the fact that there is no complete fluid structure interaction model for the simulation of the behavior of both bottom and pelagic trawls, an optimization work was started in order to settle the optimization tools and to give the initial results in this domain. For that purpose, the Finite Element Method (FEM) 3D model of the net based on triangular element developed by Priour [8], [16] is used. The triangle was chosen to describe the surface elements, so all the netting details can be represented by adjusting the triangle size. The FEM model takes into account the inner twines tension, the drag force on the net due to the current (Landweber law given above), the pressure created by the fish in the cod-end, the buoyancy and weight of the net, the mesh opening and the bending stiffness. The FEM model is able to describe the whole net and cables, which means that for a trawl, the cod-end, the wings, the headline and also the rigging up to the boat are taken into account. Triangular elements model the net while linear elements model the cables, warps and bridles. The drag and shape of both bottom and pelagic trawls can be calculated by the use of the FEM net model.

Based on this numerical model, the constrained optimization tool developed and presented in [17] and [18] starts from a reference geometry. A successive search per parameter method is then applied to find the most efficient trawl in terms of energy consumption. This automatic optimization tool can be used for both pelagic and bottom trawls but a specific optimization target should be applied in each case, being careful not to decrease the quantity of fish caught by year. The fuel consumed per year is directly linked to the drag by the towing distance by the swept width for bottom trawl and by the mouth surface for pelagic trawls. In order to decrease the fuel per catch, the objective functions used are then: the drag over swept width ratio for a bottom trawl (Figure 11) and the drag over the mouth surface for a pelagic trawl. The drag is the total drag on the trawl (Cables, netting, floats). The swept width is the mean value of the top horizontal opening and bottom horizontal opening. The mouth surface is the surface of the trawl projected onto the plane normal to the towing speed.

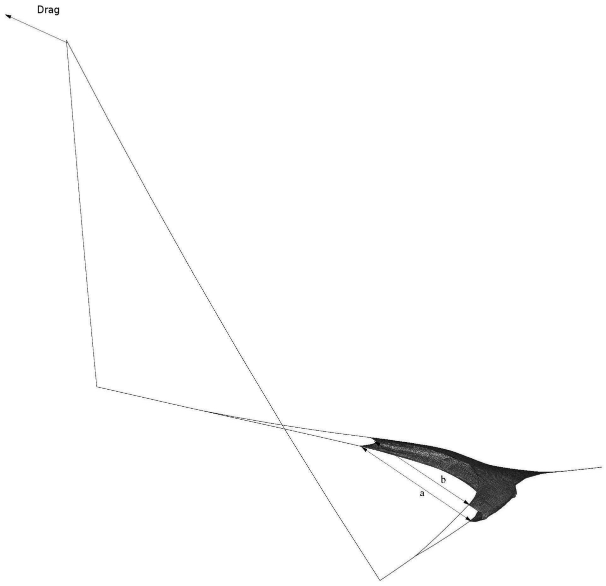

Figure 11. The objective function is the drag divided by the mean value of horizontal openings ((a+b)/2).

Due to the fact that the net is the main part of the drag, the optimization concerns the netting parts. The cables, floats and dead weights remain constant during the optimization process. So, the two main numerical control parameters of the optimisation process are the discretisation and the modification size of each net panel. The discretisation size determines the size of the triangular elements used to model the net, while the modification size determines the number of meshes each panel vertex is modified by the process. The influence of these two numerical parameters have been analysed in [8].

Applied to a bottom trawl used on research vessel (presented in [15]), the use of the optimisation tool gives an improvement of 17% on the fuel consumption (see shape comparison on Figure 12). The swept width of the optimised trawl is larger (27 m) than the reference one (22.3 m). That means a potential increase of fishing catch. In order to keep the same quantity of fish caught during a year, the swept area remains constant to the reference value by a decrease in the number of fishing trips (from 260 days per year to 215).

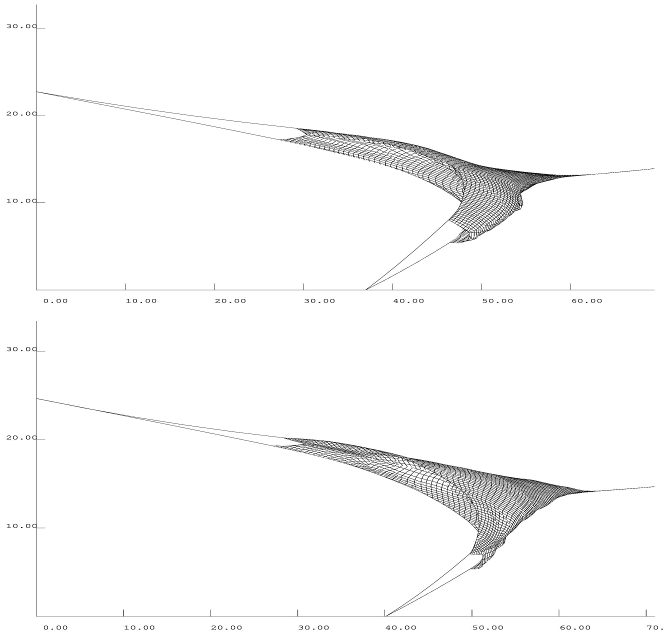

Figure 12. 3D aspects of the reference bottom trawl (top) and the optimised one (bottom). Only 1 twine out of 10 are drawn..

Applied to a pelagic trawl used for scientific surveys (presented in [19]), the use of the optimisation tool gives an improvement of 39% on the fuel consumption (see shape comparison on Figure 13). A homothetic transformation was used here in order to conserve the same mouth surface of the reference trawl (close to 200 m^2) in order to maintain the same catch a year.

In the method described above, the fishing gears have been optimized in terms of drag per swept width or per mouth surface. It is obvious that, in the mouth of the trawl, some modifications on the rope length (head-rope and foot-rope) can have a big effect on the swept area or the swept width. It may be planned to adjust automatically the length of the cables in order to optimize the gear. In fact, a lot of scientific works have been devoted to optimisation, but not for fishing gears. Such works define standard methods that could be applied to trawl optimisation.

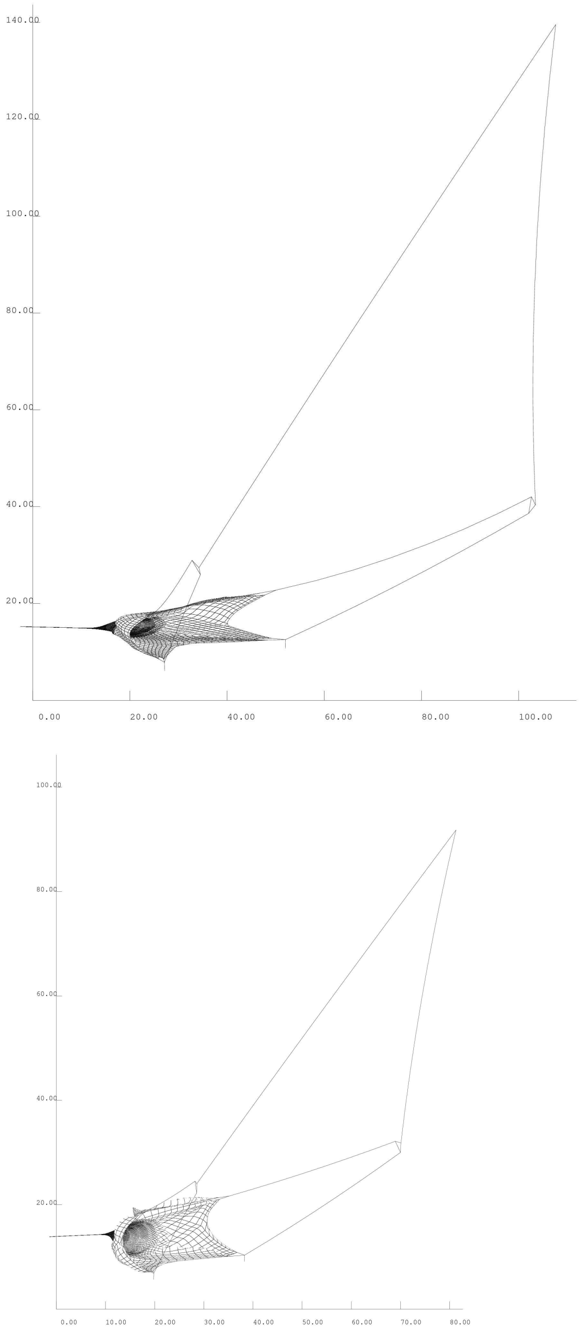

Figure 13. Design of the reference pelagic trawl (top) and the optimized one (bottom). The optimization has modified mostly the mouth of the trawl.

V. CONCLUSION

This paper presents the HydroPêche project conducted in order to develop a tool for an automatic trawl optimisation in order to find more efficient trawls in terms of energy consumption. For that purpose, different aspects have been studied in order: a/ to extend the basis of experimental data on flow characteristics governing the hydrodynamic behaviour of different porous structures ; b/ to develop numerical tools to simulate more a realistic flow around porous structures taking into account fluid / structure interactions; c/ to develop automatic optimisation tools to design efficient trawls in terms of energy consumption.

Initial results have been achieved and presented: some specific data analysis were carried out from PIV measurements around a rigid cod-end and a 1/10 scale bottom trawl, FreeFemm 3D has been used to simulate the flow around a specific cod-end and the first developments of an automatic optimisation tool give good numerical results in terms of energy efficiency for both bottom and pelagic trawls. The automatic optimisation of bottom trawl panel cutting in order to decrease the fuel consumption is based on a finite element method model adapted to fishing net structures, through a constrained optimisation tool that starts from a reference model and selects the best result according to the drag over swept width ratio. We show in the sequel that this tool offers potential saving in fuel consumption since it reduces drag. Moreover it leads to a moderate increase of catch volume while decreasing the number of fishing trips.

More accurate results should be obtained once the complete fluid/structure model to simulate the behavior of both bottom and pelagic trawls is developed.

Acknowledgment

The author would like to acknowledge and thank the following people for their implication in the project: B. Allessandrini, E. Bouhoubeiny, M Dunseath-Terao, A. Gence, R. Khaled, I. Mnasri, C. Ody, JV. Facq and B. Gaurier. The R. Lewandowski works are partially supported by the ANR Project FF2a3.

REFERENCES

[1] Ward, N., Montgomerie, M., Lart,W., 2005. Fuel efficiency trials using Jackson trawls with reduced twine diameter on MFV Challenge II Seafish Report No. SR578, p.31.

[2] Parente J., Fonseca P., Henriques V. and Campos A., 2008. Strategies for improving fuel efficiency in the Portuguese trawl fishery, Fisheries Research 93 117–124.

[3] Fredheim A., 2005. Current Forces on Net Structures, PhD Thesis.

[4] Bouhoubeiny E., Germain G., Druault P., 2010. Experimental analysis of the characteristics of the flow around bottom trawls, E-Fishing.

[5] Pichot, G., Germain, G., Priour, D., 2008. On the experimental study of the flow around a fishing net, European Journal of Mechanics B/Fluids.

[6] J.L.Lumley. “The structure of inhomogeneous turbulent flows. Atmospheric Turbulence and Radio Wave Propogation,” Vol. In A.M.Yaglom and Tararsky, pp. 166-178, 1967.

[7] Holmes P., Lumley J.L. and Berkooz G., “Turbulence, Coherent Structures, Dynamical Systems and Symmetry,” Cambridge Monographs on Mechanics, Cambridge, 1996.

[8] Le Dret H., Priour D., Lewandowski R., Chagneau F., 2004. Numerical simulation of a cod end net, Journal of Elasticity.

[9] O’Neil F.G, 1999. Axisymetric trawl cod-ends made from netting of a general mesh shape, IMA J. Appl. Math..

[10] Hunt J.C.R, Waray ,CCA, Moin P, “Streams and, Convergence zones in turbulent flows, ” center for turblence Research report CTR-S88, Stanford University, pp. 193-207, 1988.

[11] C H K Williamson “Vortex dynamics in the cylinder wake,” Review of Fluid Mechanics, Vol. 28, pp. 477-539, 1996.

[12] Vincent B., 1996. Etude expérimentale des écoulements quidés par une paroi perméable axisymétrique, PhD Thesis.

[13] Lewandowski R., Pichot G., 2007. Numerical simulation of water flow around a rigid fishing net, Comp. Methods in Appl. Mech. And Eng..

[14] Pichot G., 2007. Modélisation et analyse numérique du couplage filet-écoulement hydrodynamique dans une poche de chalut, PhD Thesis.

[15] Hecht F., Pironneau O., Le Hyaric A., Ohtsua K., 2006. FreeFem++ Manual, Laboratoire Jacques Louis Lions.

[16] Priour D., 2009. Numerical optimisation of trawls design to improve their energy efficiency, Fisheries Research.

[17] Khaled R., Priour D., Numerical method for energy optimisation of bottom trawl, E-Fishing Congress.

[18] Priour D., Khaled R., 2009. Comparison between two methods of trawl optimisation, IMAM Congress.

[19] Massé J., Koutsikopoulos, C., Patty, W., 1996. The structure and spatial distribution of pelagic fish schools in multispecies clusters: an acoustic study. ICES Journal of Marine Science 53, 155–160.