A Numerical Investigation for Underwater Fluid-Netting Interaction Problem

Ilyes Mnassri1,2 , David Le Touzé1, Benoît Vincent2, Bertrand Alessandrini1
1 Fluid Mechanics Lab., Ecole Centrale Nantes / CNRS, Nantes, France
2 IFREMER, Lorient, France

Abstract

The present work investigates an application-oriented issue in the marine and fisheries researches domain. However much trawl structure codes that simulate dynamic behavior of submerged nets, in continuous interaction with surrounded ocean water, still need more accuracy and precision due to the fact that the sea water is mainly considered as a fluid with a uniform velocity and intensity, and so the effects of small turbulence rates were neglected. When this little inaccuracy occurs in the first step of the net displacement prediction, a large error could manifests in the form of the simulated net, and as consequence, drag forces wrongly estimated. In the purpose to overcome the lack of data information exchange during the fluid structure interaction, a computational fluid program was proposed to ensure the desired coupling. The described CFD code is based on explicit finite volume method using the method of characteristics for compressible liquid flow. The objective of this CFD package is to give a quantitative description for the entire flow filed in terms of velocity and pressure value around the knots of a submerged net structure. The discretization was performed with the finite volume method for inviscid part, and a central difference approximation for dissipation terms. A Riemann solver based on Tait equation of state and implemented into Godunov-type scheme with higher order MUSCL has been used to evaluate fluxes on control volumes interfaces. In order to verify the validity of the numerical method many tests have been carried out for multi-dimensional shock tubes, open channel fluid flow and for water flow around obstacles. The goal within this work is to achieve a coupling between the net structure and surrounding fluid through the replacement of the real net obstacle, composed of thin cylinders and spheres, by force bodies, and update the fluid package to a three dimensional model. This work will lead to a more accurate computation of the Morison force displacement of the submerged net trawl.

Keywords: *net deployment*, *Navier-Stokes, Tait* EOS, *Riemann* solver

Introduction

The hydrodynamic behavior related to porous structures of fishing nets is still misunderstood. In this context, our goal is to analyze and investigate the behavior of submerged nets in the sea water in order to ameliorate the future conception of trawls and improve energy efficiency through two ways: minimizing the total energy consumption in fishing operations and avoiding catching of non- targeted species. As for the reduction of energy consumption the idea focus on reducing the drag of the net, In response, IFREMER"Institut Francais de Recherches et Exploitation de la Mer", in a cooperative partnership with many partners from academic and industrial fields, has undertaken the Hydropeche project [8] that aims to investigate the techniques concerning the flow field measurements for different types of porous fishing net structures using time-resolved PIV investigations [9], to create an automatic optimization tool that helps in the phase of trawl conception [7], and to design a numerical fluid dynamic tool to approach the real behavior of the sea water flow around the submerged net in order to be coupled to IFREMER's structure software as a routine to ameliorate the degree of computational accuracy of the net trawl displacement. The present paper deals with the third point, that is building an original simulation model for calculating net trawls behavior based on the coupling between a fluid dynamic model and a structure model through viscous body forces and net trawl displacement. The nearly viscous compressible flow field was initially modeled without body forces, and using an explicit method for time marching. To model the compressible equations, a Riemann solver with finite volume method and high order reconstruction, was introduced as a classical tool to simulate the problem and compute with a good precision the updated values of the density, pressure and velocity flow field [3]. The viscous fluxes are then discretized using central difference scheme. Computations shown begin with the consideration of a perfect gas model to validate the solver algorithm against state of the art results [1, 4]. In a second step, viscous terms and Tait equation of state (EOS) are considered for simulating water [5]. The present paper is set out as follows: introducing the basic governing equations, the numerical time and space discretization, the MUSCL reconstruction techniques, the methodology of the present Navier-Stokes solver and ends with some numerical tests. The objective of this CFD package is to give a quantitative description for the entire flow field in terms of velocity and pressure value around the knots of the submerged net structure. The resolution is performed by a finite-volume discretization for the inviscid part, and the viscous term is approximated by central differences. As for the inviscid part conservative variables are considered and the Riemann solver is implemented into a Godunov type numerical method with the higher order MUSCL scheme. This is used in conjunction with the Tait equation of state which bears as the more suitable EOS to represent the sea water characteristics. In order to verify the validity of the numerical method, different academic test cases have been investigated, first for a compressible flow (two dimensional shock tube with perfect gas), and then for a quasi incompressible viscous flow using Tait EOS in presence of a cylindrical obstacle (see fig.1 below). Good agreement with reference solutions was found on these test cases. Then Morison body forces have been added to the model to start representing the net presence. A 2D coupling is under implementation. The goal behind this work is to achieve the coupling with the 3D trawl net deployment code (see figure 1 below). The next step will thus be to realize the modeling of the 3D net structure through Morison body forces. In a final step, and as the IFREMER structure code is working with a stationary fluid flow data, the CFD solver will be linked to it as a complementary routine for fluid velocity computation, so this will lead to a more accurate computation of the Morison force displacement of the submerged trawl net.

2 Fluid modeling

2.1 Basic governing equations

The dynamics of viscous fluids is in fact purely a matter of the propagation of signals. There are two kinds of signals: sound waves and fluid movements. One can describe the entire dynamics of the fluid in terms of these kinds of signals. For 1-D fluids one can make diagrams of these signals in the (x,t) plane, and the signals propagate along lines called characteristics. For the class of finite-volume difference schemes for systems of hyperbolic conservation laws, a special solution procedure is used to compute flux vector at the cell interface in accordance with initial discontinuous data and the hyperbolic nature of governing equations. This procedure is referred to as in general to be the Riemann solver, it consist and interpreted as a technique for obtaining solution to the certain initial value problem IVP with discontinuous initial conditions. The solution obtained for one or a multidimensional case is interpreted as the evolution in time of the initial distribution. The general Navier-Stokes equations (1) describe the fluid flow motion and relate the velocity to the viscosity with the body forces.

$$\frac{\partial\left(\rho\vec{V}\right)}{\partial t}+div\left(\rho\vec{V}\otimes\vec{V}-\sigma\right)+\vec{\nabla}P=\rho\vec{f}+J \tag{1}$$

The Tait EOS [6] is suitable for describing liquid at high pressure and flows undergoing non-isentropic process, such as shock waves, similar EOS for a liquid can be found in Reference [5]. The equation (2) presents the relation between pressure and density for the Tait EOS.

$$P=\beta\left(\left(\tfrac{\rho}{\rho_0}\right)^{\gamma}-1\right)+P_0 \tag{2}$$

The sound speed c is computed as follows: $\rho c^2=\gamma\left(P+\beta\right)$

The earlier fluid flow relation (1) could be written in conservative form as:

$$\frac{\partial \Phi}{\partial t}+\frac{\partial \Psi(\Phi)}{\partial x}+\frac{\partial \mathrm{T}(\Phi)}{\partial y}=J \tag{3}$$

The set of source terms J, primitive variables Γ, conserved variables Φ and corresponding fluxes Ψ and T are given by:

$$\Phi=\begin{pmatrix}\rho\\ \rho u\\ \rho v\end{pmatrix} \qquad \Gamma=\begin{pmatrix}\rho\\ u\\ v\end{pmatrix} \tag{4-1}$$

$$\Psi=\Psi^{conv}+\Psi^{viscouc}=\begin{pmatrix}\rho u\\ \rho u^2+P\\ \rho uv\end{pmatrix}+\begin{pmatrix}0\\ -\sigma_{xx}\\ -\sigma_{xy}\end{pmatrix} \tag{4-2}$$

$$\mathrm{T}=\mathrm{T}^{conv}+\mathrm{T}^{viscouc}=\begin{pmatrix}\rho v\\ \rho uv\\ \rho v^2+P\end{pmatrix}+\begin{pmatrix}0\\ -\sigma_{yx}\\ -\sigma_{yy}\end{pmatrix} \tag{4-3}$$

$$J=\begin{pmatrix}0\\ f_x\\ f_y\end{pmatrix} \quad \text{with} \quad \left(f_x, f_y\right) \text{ the body forces} \tag{4-4}$$

The shear stresses are expressed in terms of the derivations of the velocity components (u,v) and fluid viscosity μ.

$$\sigma_{xx}=\frac{2}{3}\mu\left(2\frac{du}{dy}-\frac{dv}{dx}\right) \quad \sigma_{xy}=\sigma_{yx}=\mu\left(\frac{du}{dy}+\frac{dv}{dx}\right) \quad \sigma_{yy}=\frac{2}{3}\mu\left(2\frac{dv}{dy}-\frac{du}{dx}\right) \tag{4-5}$$

2.2 Numerical discretization

This section will focus on the spatial discretization method involving both the finite volume method for Euler equation (inviscid flow) in conjunction with high order cell reconstruction at the wall interface, and the central difference for dissipation or viscous terms.

2.2.1 Finite volume method

We present the application of finite volume method to the compressible Navier-Stokes equations. The integration of (eq.3) over a rectangular cell Ω_{ij} gives:

$$\iint_{\Omega_{ij}}\frac{\partial \Phi}{\partial \tau}+\iint_{\Omega_{ij}}\left(\frac{\partial \Psi(\Phi)}{\partial x}+\frac{\partial \mathrm{T}(\Phi)}{\partial y}\right)=\iint_{\Omega_{ij}} J \tag{5}$$

$$\frac{\partial \Phi}{\partial \tau}\Delta x\Delta y=\int_{j-\frac{1}{2}}^{j+\frac{1}{2}}\int_{i-\frac{1}{2}}^{i+\frac{1}{2}}\left(\frac{\partial \Psi(\Phi)}{\partial x}+\frac{\partial \mathrm{T}(\Phi)}{\partial y}\right)dxdy+J\Delta x\Delta y \tag{6}$$

$$\frac{\partial\Phi}{\partial\tau}=\frac{\left(\Psi^{conv}_{i-\frac{1}{2}}-\Psi^{conv}_{i+\frac{1}{2}}+\Psi^{viscouc}_{i+\frac{1}{2}}-\Psi^{viscous}_{i-\frac{1}{2}}\right)}{\Delta x}+\frac{\left(\mathrm{T}^{conv}_{j-\frac{1}{2}}-\mathrm{T}^{conv}_{j+\frac{1}{2}}+\mathrm{T}^{viscouc}_{j+\frac{1}{2}}-\mathrm{T}^{viscous}_{j-\frac{1}{2}}\right)}{\Delta y}+J \tag{7}$$

The viscous terms presents the gradient of resistance degree between adjacent molecules. The elements in the dissipation terms can be approximated by central difference scheme, therefore the derivations of the shear stresses are expressed in first order difference as below ($\Theta\equiv u,v$):

For the x direction:

$$\left.\frac{\partial\Theta}{\partial y}\right|_{i+\frac{1}{2},j}=\frac{\Theta_{i+1,j}-\Theta_{i,j}}{\Delta x} \tag{7-1}$$

$$\left.\frac{\partial\Theta}{\partial y}\right|_{i+\frac{1}{2},j}=\frac{\Theta_{i,j+1}+\Theta_{i+1,j+1}-\Theta_{i,j-1}-\Theta_{i+1,j-1}}{4\Delta y}$$

For the y direction:

$$\left.\frac{\partial\Theta}{\partial y}\right|_{i,j+\frac{1}{2}}=\frac{\Theta_{i,j+1}-\Theta_{i,j}}{\Delta y} \tag{7-2}$$

$$\left.\frac{\partial\Theta}{\partial x}\right|_{i,j+\frac{1}{2}}=\frac{\Theta_{i+1,j}+\Theta_{i+1,j+1}-\Theta_{i-1,j}-\Theta_{i-1,j+1}}{4\Delta x}$$

2.2.2 MUSCL-Hancock reconstruction

The MUSCL or the Monotone Upstream-centered Schemes for Conservation Laws was introduced by Van leer [6], this technique achieve a first order finite volume scheme, by replacing the constant piecewise value of the cell variable by a linear slope to define a newer left and right states for each cell in order to compute the inter-cell fluxes. This method faces an increasing of numeric diffusion through discontinuities and induce a lost of accuracy. The solution to this problem comes with using higher-order schemes with a locally correction of the variables used for the computation of the fluxes in such a way that the monotonic character of the solution is preserved. This technique is called Total Variation Diminishing (TVD) scheme. The following algorithm known as the MUSCL Hancock method achieve second order extension for one dimensional model. Reconstruct the conservative variable within the computational cells using the PLM (Piece Wise Linear Method) technique. This yields to new values in each computational cell using a slope limiter.

$$\Phi_i^{s,L,R}=\Phi_i^s\pm\frac{\Delta x}{2}\zeta_i \tag{8}$$

where $\zeta_i=\min\mathrm{mod}\left(\Phi_{i-1}-\Phi_i,\Phi_{i+1}-\Phi_i\right)$

Evolution of the boundary extrapolated values by a time $\Delta\tau/2$:

$$\Phi_i^{s+1/2,L,R}=\Phi_i^{s,L,R}-\frac{\Delta\tau}{2\Delta x}\left(\Psi\left(\Phi_i^{s,R}\right)-\Psi\left(\Phi_i^{s,L}\right)\right) \tag{9}$$

Riemann problem solution is found using the new conserved extrapolated values: $\Phi_i^{s+1} = \Phi_i^s - \frac{\Delta\tau}{\Delta x}\left(\Psi\left(\Phi_i^{s,R},\Phi_{i+1}^{s,L}\right) - \Psi\left(\Phi_{i-1}^{s,R},\Phi_i^{s,L}\right)\right)$ (10)

The Riemann solver to compute the flux will be discussed in later section. Remark that this reconstruction concerns only the convective terms.

2.2.3 Navier-Stokes solver

The resolution of the complete compressible Navier-Stokes is divided into three steps. First the computation of convective fluxes at interfaces with a Riemann solver throughout a Godunov numerical type scheme couched in terms of finite volume discretization, next step comes with the addition of the viscous fluxes, and finally once we updated the conserved variable, it will be used to integrate the partial differential equation for the source term.

2.2.4 Godunov scheme

The Godunov approach aims to provide a solution by including the possible discontinuous character of the solution directly in the numerical method. Godunov-type methods consider the numerical solution as being discontinuous in essence, a continuous profile being a particular case of a discontinuous one. In these methods, space is discretized into volumes, or cells, so the general term of finite volume method. The numerical solution is not characterized by its value at points, but by its average over the cells. The exchange via fluxes determines the evolution of the solution in a given cell at the interfaces with all the neighboring cells. In the Godunov approach, the fluxes are computed by solving Riemann problems at the interfaces between the cells. The powerful character of this approach is that the exchanges between the fluxes can be computed if the solution is initially discontinuous.

2.2.5 Source term

The discretization of the complete Navier-Stokes equations containing the source terms (eq.6) is subject to a different way of resolution. One common way is to resolve the homogenous problem with only viscous and convective terms, then using its solution as an initial value for the partial differential equation.

$$\begin{cases} \dfrac{\partial\Phi}{\partial t} + \dfrac{\partial\Psi(\Phi)}{\partial x} + \dfrac{\partial\mathrm{T}(\Phi)}{\partial y} = 0 \\ \widetilde{\Phi} = \Phi + \Delta\tau\left(\dfrac{\Delta\Psi}{\Delta x} + \dfrac{\Delta\mathrm{T}}{\Delta y}\right) \end{cases} \tag{12}$$

$$\frac{\partial\widetilde{\Phi}}{\partial\tau} = J \tag{13}$$

2.2.6 Time integration

The equation (12) is integrated with respect to time with an explicit 2nd order Runge-kutta (*R-K*) method. This method guarantees time stability and good accuracy. The purpose behind the explicit time integration choice was the ability of achieving a simple coupling for our present hydrodynamic solver with the mechanical structure code. Due to the higher computer performance and it's massively parallel architecture, the main disadvantage of long computer running times detected in explicit method is covered. The Runge-kutta procedure is described below:

$$RK_1 = \Delta\tau J\left(\tau^s, \Phi^s\right)$$

$$RK_2 = \Delta\tau J\left(\tau^s + \Delta\tau, \Phi^s + RK_1\right)$$

$$\Phi^{s+1} = \Phi^s + \frac{\left(RK_1 + RK_2\right)}{2} \qquad (14)$$

Here s and $s+1$ refers to the time interval, whereas RK_1 and RK_2 presents the *R-K* stages.

2.2.7 Numerical results

The current compressible Navier-Stokes code was successfully tested for several one and two dimensional geometries, for both gas dynamics and hydrodynamic cases, in what follows we will presents some the validation results. The first test was accrued on the two-dimensional gas shock tube test with symmetric boundary conditions to validate the core of the code (fig.5), the results compares well with Tadmor and Kurganov [4] in terms of density contours. In order to test the inlet outlet boundary conditions for open channel, the code was applied to the Mach 3 wind tunnel problem (fig.6) mentioned in the works of Woodward and P. Collela [6] and the results prove the efficiency of the designed code to capture discontinuity and compute well the fluxes at boundaries. Once we get good agreement with gas dynamics we move to the water flow and look to validate the code with an equation of state for water. Figure (7) shows the development of velocity contours and streamlines behind an obstacle for an open water flow channel [14]. Velocity profiles Figures (8-9) are compared with those obtained by Pengzhi [15]. The whole results prove the efficiency of the hydrodynamic solver. A good accuracy has been observed and a satisfactory progression could be done for the coupling phase with the mechanical code.

3 Structure code

The IFREMER structure code [10] is designed to predict the net deployment considered as a set of rigid bars. In general this program uses the fundamental principle of dynamics combined to elasticity equations presenting the bars. A general overview of the trawl is presented in (fig.2). The hydrodynamics forces acting in the problem are expressed using the Morrison hypothesis. In this section we present briefly the set of equations in general forms. The Mechanical equations applied to each knot of the net presents the sum of all external forces is equal to half of the mass of all rigid elements around each application point times the acceleration of the coordinate displacement. The following equation (15) describes the acting forces in the problem [11, 12].

$$\sum_{j=n_1(i)}^{n_k(i)} \left(X_{ij} + \frac{(Y_{ij} + R_{ij} + Z_{ij})}{2} + B_{ij} \right) = m_{ij} \frac{d^2 \ell}{d\chi^2}$$

Here we design by k the number of knots adjacent to knot i, $n_1(i),....n_k(i)$ the numbers of the knots adjacent to knot i, X_{ij} is the tension of bar ij, B_{ij} the linking forces applied on the each knot (fig.3).

The hydrodynamic forces acting on the bar ij are expressed as follows:

The drag pressure force orthogonal to the element: $Y_{ij} = -0.5\rho dlC_d V_P \|V_P\|$ (16)

The tangential friction force: $R_{ij} = -0.5 f \rho dlC_d V_t \|V_t\|$ (17)

The added mass: $Z_{ij} = -0.25\rho d^2 \pi lC_m \gamma_P$ (18)

4 Fluid Structure interactions

The problem of fluid structure interaction investigated in this work describes the linking between a fluid model and a mechanical structure code. The major idea here is to replace the flexible net structure, modeled by a set of cylindrical wire and spherical knots and facing the sea water flow by source term forces from Morrison equations, by this modification in the manner of considering the net obstacle we would avoid a huge amount of computational time. In the second section the forces acting in the mechanical code have been presented and defined, the goal being to achieve is to get more accurate hydrodynamic forces throughout the correction of fluid velocity values in the Morrison hypothesis. The idea to link the present work to the IFREMER program will be performed in two steps; the first step is to begin calculation with structure code to have the initial hydrodynamic forces values, next step is to replace the body forces in the fluid code by this term sources. This will affect the values of the fluid velocity field and arise in more accurate Morrison forces computations. The future work to be completed will involve the migration to three dimensional version of the present program and the linking of the hydrodynamic code, the major objective being to improve the conception of next generation of trawl nets. The primary results of the linking with source terms show the fluctuation of velocity field behind the application points where we put source terms expressing body forces. Figure 8 exhibits that current calculations are encouraging and follow the expected velocity fields.

5 Concluding remarks

In order to simulate sea water flow around trawl nets, a c++ code has been designed to resolve the weakly compressible Navier Stokes equations in conjunction with the Tait equation of state. The algorithm combines high-order MUSCL reconstruction and Runge Kutta time integration. This general purpose code has been validated on different applications including both gas and hydro dynamics problems. A correlation between source term force and obstacle diameter is under investigation to achieve a good agreement of results through replacement of obstacle by hydrodynamic force source terms. However a parallelization and local refinement around source terms will be done for better software performance.

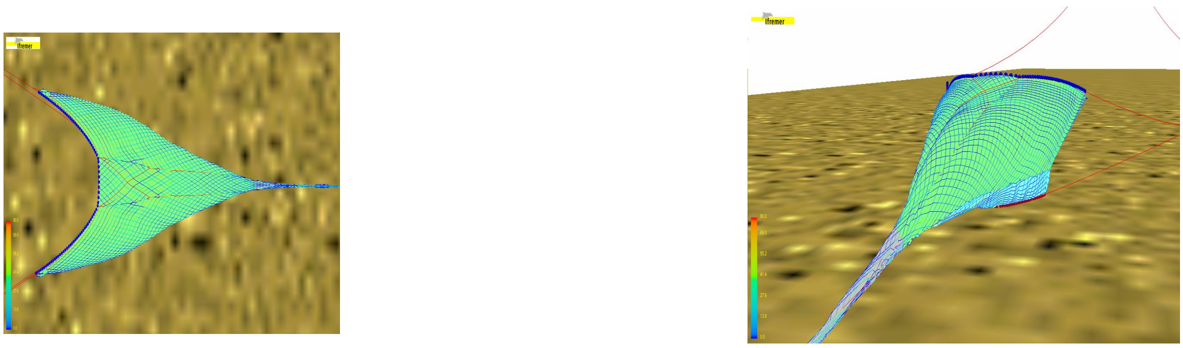

Fig 1 : IFREMER net structure deployment

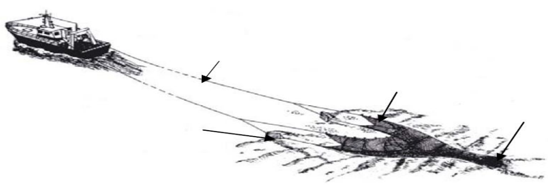

Fig.2 A General overview of a trawl [13]

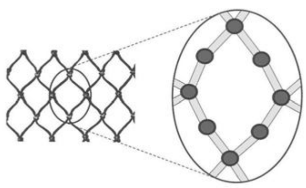

Fig.3 Representation of the net meshes [11]

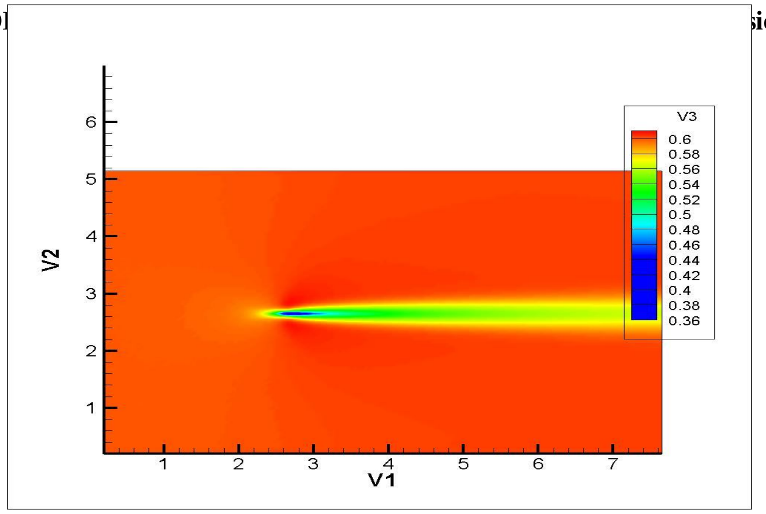

Fig.4 Velocity contours in the presence of source term

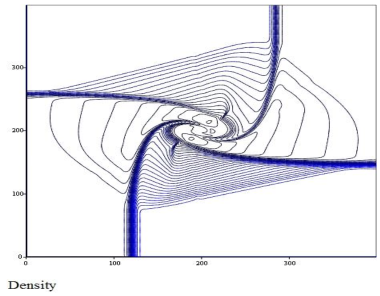

Fig.5 Density contours for 2D shock tube

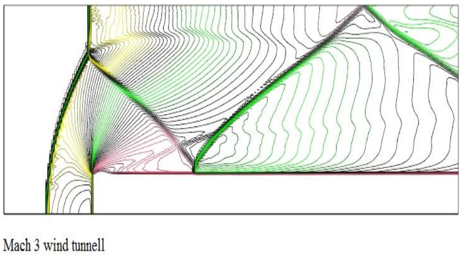

Fig.6 Mach 3 wind tunnel

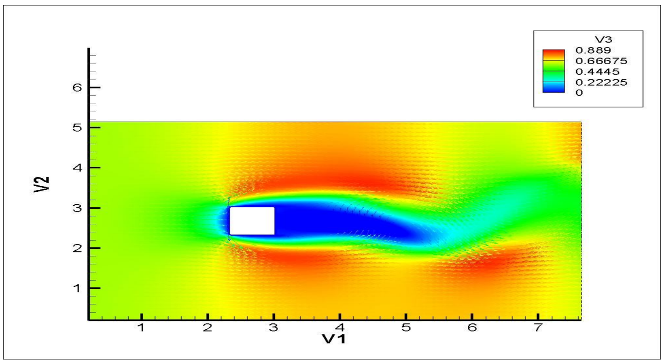

Fig.7 Velocity contours for Re = 150

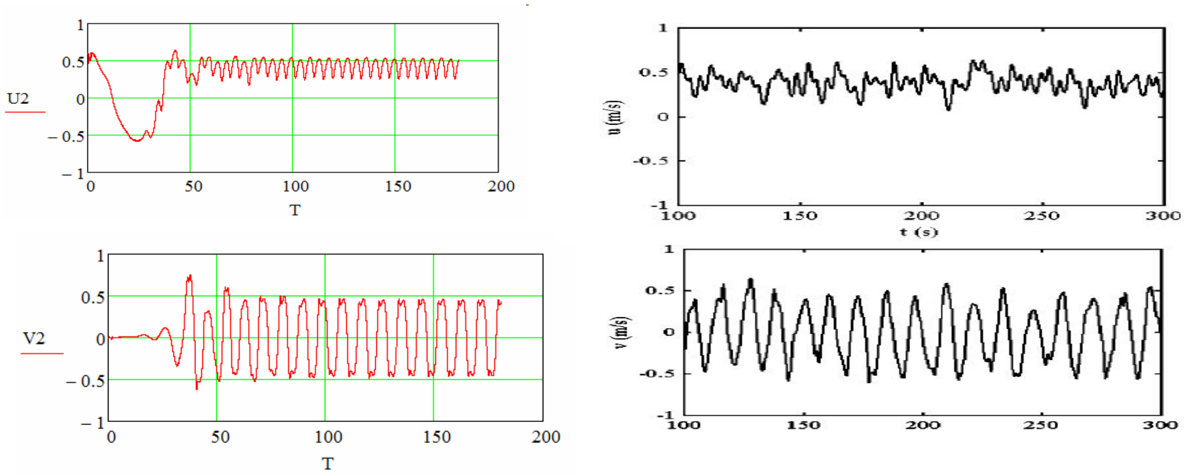

Fig.8 Velocity profile

References

1. A. Jameson, W. Schmidt, and E. Turkel. Numerical Solution of the Euler Equations by Finite Volume Methods using Runge-Kutta Stepping Schemes. Technical Report 81-1259, AIAA, 1981.

2. Tze-Jang Chen and C.H. Cook A New Averaging Scheme for the Riemann Problem in Pure Water. Math. Comput. Modelling, 25(3) 25-36 (1997)

3. E.F.TORO Riemann Solvers and Numerical Methods for Fluid Dynamics. Springer, Berlin, (1997)

4. A. Kurganov and E. Tadmor Solution of Two-Dimensional Riemann Problems for Gas Dynamics without Riemann problem solvers. Wiley periodics, (2002)

5. M. J. Ivings, D. M. Causon, and E. F. Toro. On Riemann Solvers for Compressible Liquids. Technical Report 97-4, Department of Mathematics and Physics, Manchester Metropolitan University, UK, 1997.

6. P. Woodward and P. Collela. The Numerical Simulation of Two-Dimensional Fluid Flow with Strong Shocks. J. Computational. Physics . 54, 115–173 (1984).

7. Daniel Priour Numerical optimisation of trawls design to improve their energy efficiency http://archimer.ifremer.fr/doc/2009/publication-6583.pdf

8. Grégory Germain , Philippe Druault , Roger Lewandowski , Benoit Vincent , Daniel Priour , Jean-Yves Billard HydroPêche: a way to improve energy efficiency of fishing devices First International Symposium on Fishing Vessel Energy Efficiency E-Fishing, Vigo, Spain, May 2010

9. Elkhadim Bouhoubeiny, Gregory Germain, and Philippe Durault Time-Resolved PIV investigations of the flow field around cod-end net structures. Fisheries Research (2011)

10. http://www.ifremer.fr/dynamit/en/

11. Study of the manoeuvrability and security of a trawl gear Marichal Dominique, Benoit Vincent
12. Study of dynamics of submerged supple nets (applications to trawls) J.S. Bessonneau, Marichal Dominique Ocean Engineering Vol. 25 No. 7 pp- 563-583, 1998
13. Georges, J.-P., Nedelec, C. (1991): Dictionnaire des engins de pêche, Ifremer, Editions Ouest-France
14. A. Etminan, M. Moosavi and N. Ghaedsharafi Characteristics of aerodynamics forces acting on two square cylinders in the streamwise direction and its wake patterns.
15. Pengzhi Lin Wave–current interaction with a vertical square cylinder. Ocean Engineering 30 (2003) 855–876