**Proceedings of the 6th Numerical Towing Tank Symposium. Rome, Italy, 29 September - 1 October 2003**
Pages 35-39
http://archimer.ifremer.fr/doc/00455/56631/




# Numerical study of a suction anchor

Grosjean François [1,*], Kerampran Steven [1], Rosu Octavian [1], Bompais Xavier [2]

[1] ENSIETA, F-29200 Brest, France
[2] IFREMER, Laboratoire d'hydrodynamique appliquée (TMSI-RED-HA), Plouzané, France

* Corresponding author : François Grosjean, email address : grosjefr@ensia.fr

**Abstract :**

This work aims at simulating the movement of a suction anchor using a new version of commercial code *Fluent*. The computations consider a sinusoidal movement of the anchor (roughly corresponding to its heave) both in a fluid at rest and in a uniform flow. They are validated with results of experimental tests performed by Ifremer using a scaled model of suction anchor.

## Introduction

The suction anchor is a new concept designed to provide strong mooring to offshore platforms. This anchor is a hollow steel cylinder (over 15 m in height) that is deeply embedded in the seabed. Part vacuum is created inside the anchor to ensure a good fixation. The positioning of such a cumbersome element causes some problems to arise, especially when the anchor is lowered from the ship to the seabed. During this translational vertical movement, the anchor is submitted to the ship heave. This additional movement is likely to generate extra constraint on the wire, which could in turn lead to its deterioration. Experimental tests have been performed by Ifremer, using a scaled model, Fig.1. This work aims at simulating the movement of the anchor using a new version a commercial code Fluent. The computations will consider a sinusoidal movement of the anchor (roughly corresponding to its heave) both in a fluid at rest and in a uniform flow. They will be validated with the experimental results provided by Ifremer.

## Experimental setup

Experimental results are issued from tests realised in the basin of Ifremer, at Brest, in December 2001, in partnership with Bouygues-Offshore. The basin is 10 meters depth, 50 m long and 12.5 m large, Fig.2. For these tests, a model of suction anchor is hanged on a vertical wire loop, which is anchored at the bottom by two pulleys and moved at the top by a servomotor. The servomotor is PC-controlled. . To simulate the heave of the anchor, vertical sinusoidal motions are imposed at several periods and amplitudes, with or without superposed vertical translatory motions. Sensors measure out the displacement, speed and acceleration of the model as well as the vertical hydrodynamic forces on it.

The hydrodynamic forces are supposed to be as :

$$F_z = M_a . \frac{d^2 z}{dt^2} + B . \frac{dz}{dt} . \left| \frac{dz}{dt} \right|$$

where $z$ is the vertical displacement of the model. $M_a$ and $B$ are two coefficients which characterize the hydrodynamic behaviour of the model, respectively the added mass and the quadratic damping coefficient.

A numerical identification using the Fourier series of force and displacement provides these two coefficients. The comparison between numerical and experimental results can be then more accurate.

## Model description

We use the Beta version of the code FLUENT 6 which includes a new functionality enabling to deform the mesh and therefore generate the unsteady movement of a body. This version will be tested on the simple movement of the suction anchor.

We choose a computational domain according to the experimental set up, which itself has a reduction coefficient of ten. The main geometrical characteristics of the anchor are given hereafter. Its upper surface is doted with two circular holes to allow water to flow inside the anchor. These holes can be caped if need be.

- length 1730 mm, diameter 450 mm, Fig.3
- lateral thickness 4 mm, border thickness 10 mm,
- holes - diameter 31 mm, height 80 mm.
- weight 34.4 kg,

## Meshing

Given the geometry of the anchor, it is tempting to use a 2D axisymmetrical mesh. In this case, the holes on the upper surface cannot be correctly modelled, and are substituted with a single hole on the axis of the anchor, with an equivalent area. Although this introduces some minor changes in the flow, we believe that these changes are well balanced by a considerable decrease of the mesh size and computational time. A very tight structured grid is generated, in particular in the zone close to the hole and along the anchor, to accurately compute the flow in the boundary layer, Fig.3. The number of hexahedral cells is 37000. In order to check the influence of the changes in the geometry of the upper face, a 3D hexahedral mesh has also been generated, Fig.4, using approximately 110000 cells.
To use the dynamic layering model implemented in Fluent 6, three zones are defined. One will move with the anchor and will include the dynamic mesh the two other zones, which surround the first one, will be fixed. In a given fixed zone, cell layers are either created or destroyed according to whether the dynamic zone moves towards or from a fixed zone. Several parameters can be used to monitor the creation and destruction of cell layers, including the layer thickness, Fig.5:
if the dimension of the cell is not between two given values $[c \cdot l, (1+s) \cdot l]$, the code will add a new cell layer [1]. The addition or suppression of layers occurs in the vicinity of the dynamic grid.
The parameters for this dynamic layering are:

- $c \rightarrow$ a dislocation coefficient <<split/collapse factor>> (*c=0.4*) ;
- $s \rightarrow$ a regeneration coefficient <<spring constant factor>>( *s=1) ;*
- $l \rightarrow$ reference dimension along the *x* axis (movement axis) for elementary cell (*l = 0.015m*).

Given the fact that we used a beta version, only the movement of a rod could be modelled at first. Later on we have had the possibility to build a movement using a programmable function. This point has been very important for the success of this study.

## Resolution

The fluid is incompressible but viscous. The boundary conditions are: *velocity inlet of 0.8m/s, outflow in the vicinity of the seabed, symmetry along the axis.*
The Reynolds number is $1.45 \cdot 10^6$ Generally we use the k-ε standard model for turbulence. The k-ω standard model has also been used on some occurrences.
Residuals are reduced to $10^{-4}$. We choose an implicit segregated scheme. After convergence with 'first-order' scheme, we use a second-order scheme.
We do not use the gravity because the experimental set-up eliminates its effects.

## First Numerical results

The first test was to compare the results in 3D with 2 holes and the results in the axisymmetrical case with a single hole. We did several calculations for a stationary flow and although some noticeable changes in the flow were observed, the net results for the drag on the anchor were strictly the same.
To compare with the experimental results, we choose the following sinusoidal movement: $z = A \cdot \cos(2 \cdot \pi \cdot t / T)$. The movement begins with the anchor first going down. The forces are positive when they are oriented toward the top.
Several cases have been simulated:

- period T=2.222s and several amplitudes *A=0.1, 0.15, 0.2 (m),*
- amplitude *A=0.1m* and several periods *T=1.389, 2.222, 3.03, 4.762 (s)*,
- in the case, *T=1.389s, A=0.2m*, we simulate both situations with the hole open and shut. The differences between the holes open or shut seem to be insignificant.

As can be seen on Fig.5, the forces on the anchor increase when the amplitude is increased.

Comparison with the experimental results shows a poor agreement, Fig.6. Important differences can be observed for the maximum and minimum effort (in the legend, 'mgama' denotes the force due to added mass).
This discrepancy between the numerical and the experimental results is observed in all cases, whatever the amplitude and the periodicity.

**Displacement**

The movement is given with the function $z = A \cdot \cos(2 \cdot \pi \cdot t / T)$, but in the beta version of the code we have to use the movement of a piston. When the theoretical movement: $z = A \cdot \cos(2 \cdot \pi \cdot t / T)$ (« cos » on Fig.7) is compared to the actual numerical path, it can be seen that the two curves are not superimposed. In fact, the extremes are almost identical but there is a slight difference in the medium part of the curves because the length of the piston is not large enough. The movement in the code doesn't correspond to our case. The Fourier analysis shows that the ratio L/R (length of the rod L and the diameter R of the crank) must be larger than 10 to have little influence on the symmetry of the movement.
If the length of the rod is too large, it becomes difficult to reach the convergence of the computations. By changing progressively this length from 10 to 20, this convergence is eventually obtained, and in the last step we can calculate the movement for a length of 20.
In the final version of Fluent 6, it is possible to model a real movement using a function.

**Numerical results**

When the movement of the anchor is corrected and is truly sinusoidal, the agreement between numerical and experimental results turns out to be very satisfying for all the studied cases. Some examples are shown on pictures 8 and 9, for the anchor in a fluid at rest. When we calculate the damping coefficients between the holes open or shut, both for the numerical and experimental results, we can see that when we shut the holes, this coefficient is reduced by 25%. For translation and heave, the comparison is again excellent, except close to the extreme where we have some discontinuities most likely due to a large time step. Unfortunately the time step could not be further decreased, since it took a week to obtain the results for 4 periods with a Sun Ultra 10 workstation.

**Conclusions**

Several conclusions can be drawn from this work:

- We have to be careful about the imposed movement; programming the movement with the rod isn't very convenient. The best way to create the good movement is to use the profile function in Fluent.
- The results are not affected by the opening of the holes on the upper surface of the anchor. The results are very sensitive to the movement, but if the movement is accurate, the numerical results are very close to the experimental ones.
- On this example, we consider ourselves satisfied by the new version of Fluent. We have begun to simulate other movements using programmed functions.

**REFERENCES**

[1] Documentation FLUENT version 6.0. 2002
Server FLUENT : *http://www.fluentusers.com*

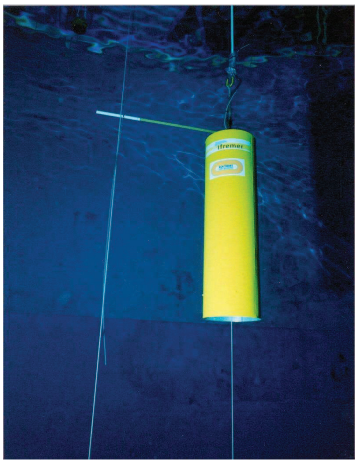


Fig.1: The experimental anchor

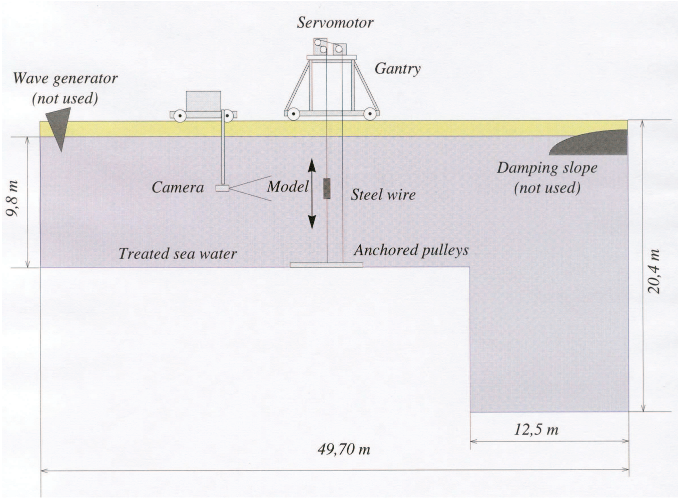
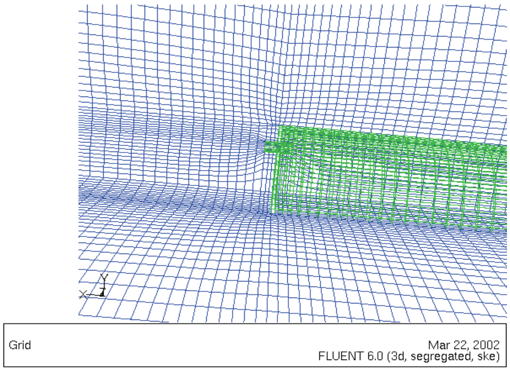
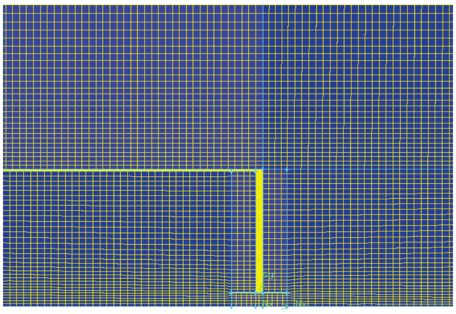



Fig.2: The experimental set-up



Figs.3 and 4: The 3d mesh and the axisymmetrical one in the vicinity of the hole

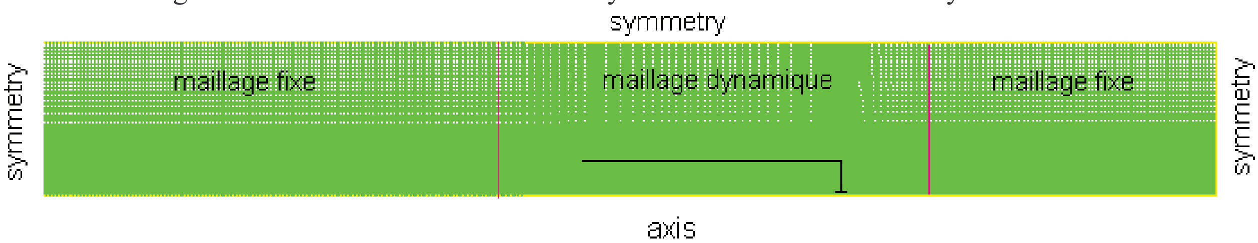



Fig.5: The grid and the boundary conditions

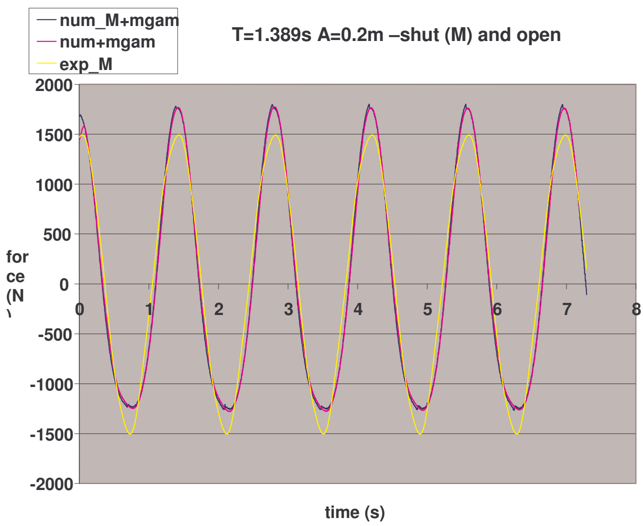



Fig6: First results

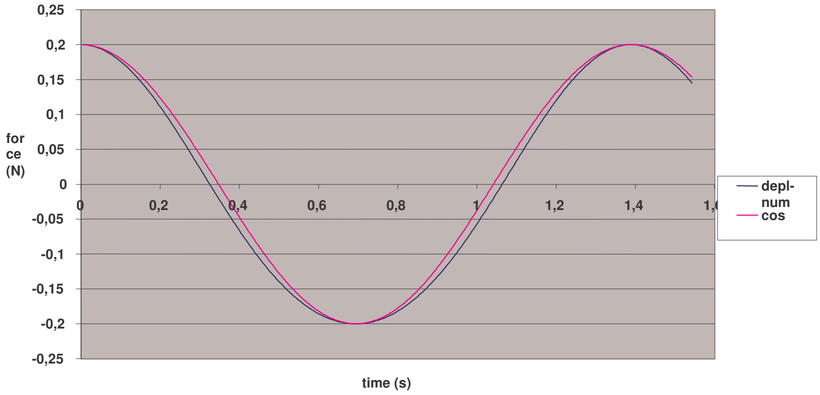



Fig.7: Experimental and numerical movements

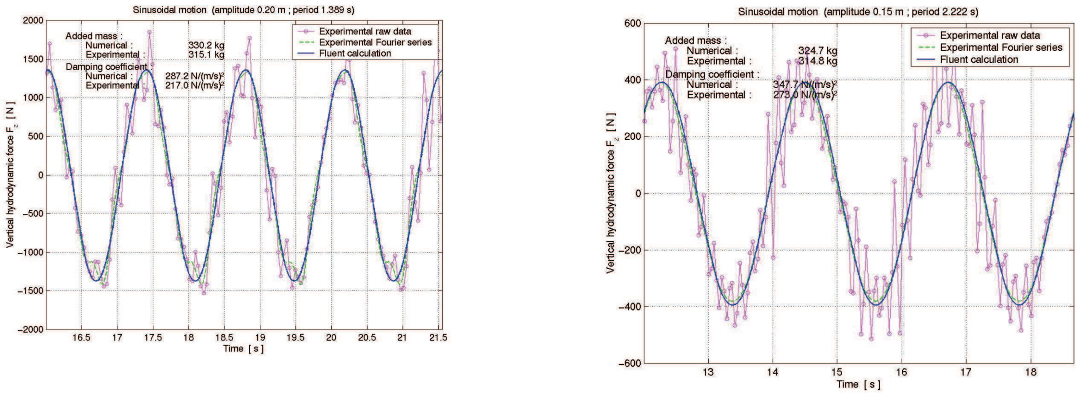



Fig.8: Comparison of experimental and numerical results for a sinusoidal movement

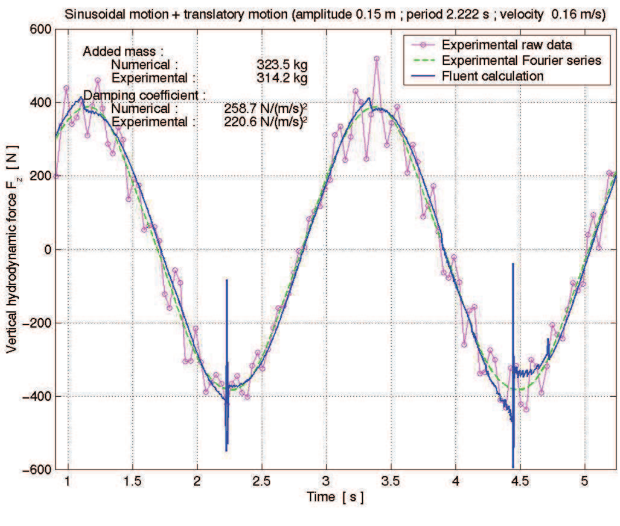



Fig.9: Comparison of experimental and numerical results for a sinusoidal movement with translation