



HAL
open science

Numerical simulation of cavitating flow around a 2D hydrofoil: barotropic approach

Benoît Pouffary, Regiane . Fortes Patella, Jean-Luc Reboud

► To cite this version:

Benoît Pouffary, Regiane . Fortes Patella, Jean-Luc Reboud. Numerical simulation of cavitating flow around a 2D hydrofoil: barotropic approach. Fifth International Symposium on Cavitation (CAV2003), Nov 2003, Osaka, Japan. hal-00211991

HAL Id: hal-00211991

<https://hal.science/hal-00211991>

Submitted on 2 Mar 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution 4.0 International License

NUMERICAL SIMULATION OF CAVITATING FLOW AROUND A 2D HYDROFOIL: A BAROTROPIC APPROACH

B. Pouffary

*Laboratoire des Ecoulements
Géophysiques et Industriels,
Grenoble, France
pouffary@hmg.inpg.fr*

R. Fortes – Patella

*Laboratoire des Ecoulements
Géophysiques et Industriels,
Grenoble, France
fortes@hmg.inpg.fr*

JL. Reboud

*CNRS-LEMD, University of
Grenoble, France
jean-luc.reboud@grenoble.cnrs.fr*

ABSTRACT

This paper presents a numerical study of the flow around a 2D hydrofoil, proposed as a test case for the CAV2003 workshop on physical models and CFD tools for computation of cavitating flows. Non-cavitating and cavitating conditions are investigated. The phenomenon of cavitation is modelled through a single fluid model, associated with an arbitrary barotropic state law.

The non-cavitating study focuses on the influence of the turbulence model and on the dependence of the results to the mesh used in the computations. The results predicted by three different turbulence models (Baldwin-Lomax, Spalart-Allmaras and k-ε with extended wall function) are compared.

The cavitating study first presents an unsteady behaviour of the partial cavity attached to the foil. A quasi-periodic shedding of vapour clouds is observed and in depth analysed. Then, an analysis of a supercavitating condition is performed.

NOMENCLATURE

c: chord length
 C_{μ} : turbulent viscosity coefficient
 f: main frequency of cavity self-oscillation
 k: turbulent kinetic energy
 P: static pressure
 P_{ref} : outlet static pressure (imposed)
 P_{vap} : vapour pressure
 U_{ref} : inlet velocity (imposed)
 V: vapour volume
 ε: turbulent dissipation
 μ_t : turbulent viscosity
 ρ: density
 ρ_l, ρ_v : liquid and vapour density

non-dimensional parameters :

α: void ratio
 l_d : reduced (by c) abscissa of cavity detachment
 l_{max} : reduced (by c) cavity length

t_{max} : reduced (by c) cavity thickness

l_{max} : reduced (by c*lmax) position of maximum cavity thickness

σ: cavitation parameter $\sigma = \frac{P_{ref} - P_{vap}}{\frac{1}{2}\rho U_{ref}^2}$

C_l, C_d : lift and drag coefficient $C_{l,d} = \frac{Force}{\frac{1}{2}\rho U_{ref}^2 c}$

C_p : pressure coefficient $C_p = \frac{P - P_{ref}}{\frac{1}{2}\rho U_{ref}^2}$

St: Strouhal number $St = \frac{fl_{max}}{U_{ref}}$

Tu: turbulent intensity $Tu = \frac{\sqrt{\frac{2}{3}k}}{U_{ref}}$

numerical parameters:

Δt: time step
 $\kappa^{(2)}$: dissipation coefficient
 NCAV: number of iterations in the transient of P_{vap}
 CFL: Courant-Friedrichs-Lewy number
 RELAX: under-relaxation coefficient of density
 ITMAX: maximum number of inner iterations

INTRODUCTION

The numerical simulation of two phase flows remains a major challenge on both physical and numerical aspects. Indeed, two phase flows, and moreover cavitating flows, involve a wide range of scales, which generates difficulties for the modelling. For that reason many approaches have been developed during the last decades. Moreover, validating a numerical and physical model necessitates experimental data that are still nowadays very difficult to obtain. In this frame, the definition of a common test case in order to compare the numerical predictions should provide a good opportunity to discuss on the modelling issues.

The approach that has been developed in Grenoble (LEGI) for many years is based on the assumption that cavitation can be modelled by considering a single fluid hypothesis and a barotropic behaviour of the mixture. A major part of this research has been motivated by the importance of cavitation as a critical phenomenon for the spatial industry. In that scope, the French space agency CNES and SNECMA Moteurs have supported research with the purpose of developing and validating a 3D N-S code allowing to simulate the cavitating operation of turbopumps. In this context, a cavitation module is being implemented in the CFD commercial code FINE/TURBO™ developed by Numeca International [1].

The present study illustrates the results we have obtained and the analysis we have performed on the CAV2003 2D hydrofoil proposed as a test case for the workshop on physical models and CFD tools for computation of cavitating flows. The non-cavitating operation was first characterised in details as a reference for cavitating conditions. Influences of the mesh and of the turbulence model were studied, mainly by comparing the values of lift and drag. Then, two cavitating configurations were separately analysed: $\sigma=0,8$ where an unsteady cavitating behaviour was obtained and $\sigma=0,4$ where a supercavitating flow was observed.

1. PRESENTATION OF THE NUMERICAL MODEL AND OF THE TEST CASE

1.a) Physical model

In the present work, we apply a single fluid model based on previous numerical and physical work developed at LEGI [1-3]. It considers the liquid-vapour mixture as a single fluid, characterised by a density ρ that varies in the computational domain according to a state law. When the density in a cell equals the liquid one ρ_l , the whole cell is occupied by liquid, and if it equals the vapour one ρ_v , the cell is full of vapour. It is worth noting here that what is called in the model “vapour density” is a numerical vapour density, the physical one being so low that it generates numerical instabilities. Consequently a liquid to vapour density ratio of 100 will be considered here. Between these two extreme values, the cell is occupied by a liquid/vapour mixture that we consider as one homogeneous single fluid.

Moreover, our model assumes that locally velocities are the same for liquid and for vapour: in the mixture regions vapour structures are supposed to be perfectly carried along by the main flow. This hypothesis is often assessed for this problem of sheet-cavity flows, in which the interface is considered to be in dynamic equilibrium [4-6].

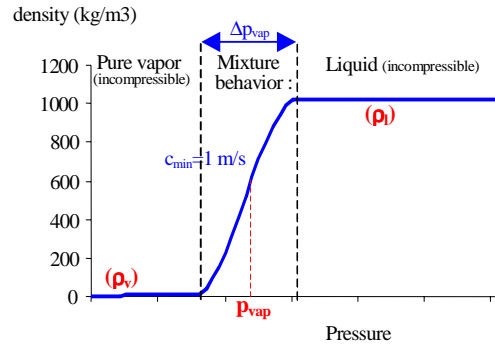


Figure 1: the barotropic state law $\rho(P)$

In the present work the arbitrary barotropic law $\rho(P)$ used is presented in Figure 1. Pure liquid and pure vapour are supposed incompressible: $\rho=\rho_l$ or $\rho=\rho_v$. The two fluid states are joined smoothly in the vapour-pressure neighbourhood by a portion of sine function. It results in the evolution law characterised mainly by its maximum slope $1/c_{\min}^2$, where $c_{\min}^2 = \partial P / \partial \rho$. c_{\min} can thus be interpreted as the minimum speed of sound in the mixture. Its calibration was done in previous studies by Reboud et al. [3]. The value $c_{\min} \approx 2$ m/s is applied for the computations presented hereafter.

1.b) Numerical model

To calculate cavitating flows we apply the commercial code FINE/TURBO™ developed by NUMECA International. It is a three-dimensional structured mesh code that solves the time dependant Reynolds-averaged Navier-Stokes equations. A detailed description of the code is given by Hakimi [7].

Time accurate resolutions use the dual time stepping approach proposed by Jameson [8]. Pseudo-time derivative terms are added to the equations. They march the solution towards convergence at each physical time step. The code resorts to a multigrid strategy to accelerate the convergence in non-cavitating conditions, associated with a local time stepping and an implicit residual smoothing.

This kind of resolution is devoted to highly compressible flows. In the case of low-compressible or incompressible flows, its efficiency decreases dramatically. This well-known problem has been addressed by many authors and solved by introducing a preconditioner [7]. This one is based on the studies presented in [9-10]. It consists in multiplying the pseudo-time derivatives by a preconditioning matrix Γ^{-1} .

The discretization is based on a finite volume approach. We use a second order central scheme that must be associated with two artificial dissipation terms, respectively of second and fourth order. The first one is activated in the strong pressure and density gradient areas. The other one is used in the whole domain, and it results in a second order space accuracy. The pseudo-time integration is made by a four-step Runge-Kutta procedure.

The physical time-derivative terms are discretized with a second order backward difference scheme that ensures a second order accuracy in time.

1.c) Test case and numerical procedure

The geometry used for the calculations is the one defined analytically by the organisers of the workshop (fig. 2). Blue line is the inlet, red line the outlet and green lines, the solid walls. As boundary conditions a constant velocity profile was imposed at the inlet (6 m/s) and a constant static pressure was imposed at the outlet. A turbulence intensity of $Tu=0,1\%$ was also imposed at the inlet.

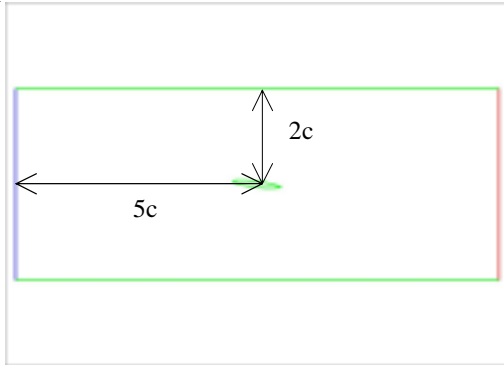


Figure 2: geometry

To simulate the cavitating flows, a non cavitating computation is initially performed. The development of cavitation is then imposed and controlled through the evolution of the vapour pressure used in the barotropic state law. This vapour pressure is increased linearly from a value insuring the absence of cavitation until the required cavitation number σ . The cavitation number is then kept constant throughout the computation.

1.d) The meshing

The first calculations were performed considering four different topologies, illustrated on fig. 2-3. The influence of the mesh was evaluated in the case of non cavitating simulations and main results are presented in the next paragraph.

2) NON-CAVITATING ANALYSIS

This section will present the methodology that has been applied to select the mesh and the model of turbulence used for the cavitating computations. The comparisons will be based on convergence criterias and predicted values of lift and drag.

2.a) Influence of the mesh

First of all, an analysis of the influence of the mesh on the non cavitating predictions was performed in two steps:

- definition of the topology

To minimise the CPU time, we first tried to create a mesh topology that would allow a coarse grid in the regions of no major interests. To achieve this goal, we made a refined C block around the foil connected to the boundary conditions through coarser blocks (fig. 3a and 3b):



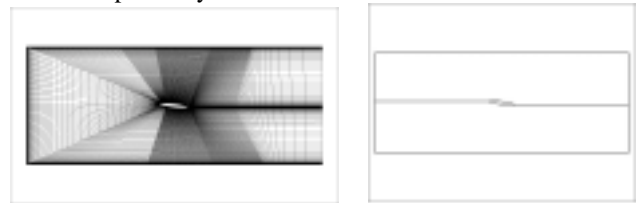
a) 40660 nodes

b) 42120 nodes

Figure 3: mesh topologies

The first configuration was rejected because of difficulties to convect turbulent quantities such as k and ϵ in the wake through the connection between the C block and the downstream block. The second configuration was found to create problems concerning the conservation of total pressure at the connection between the upstream block and the others. This numerical error was due to the skewness of the mesh.

As those trials led to failures, we decided to generate meshes without coarse blocks. Then, the topologies 4a and 4b were tested : a unique C block and a mesh composed of two H blocks respectively under and over the foil.



a) 45680 nodes

b) 71442 nodes

Figure 4: C and H mesh

As it was found that the second topology needed more cells than the first one, we decided to retain the C topology for the cavitating study. The H mesh will only be used to evaluate more precisely the influence of turbulence model on non cavitating computations as it enables computing with a multigrid strategy enhancing a convergence acceleration.

- definition of the refinements

Considering the C topology presented before, two different meshes have been realised. They are respectively composed of $60*443$ (mesh 1) and $80*571$ (mesh 2) cells. For C and H meshes, the distance between a solid boundary and the first cell equals $50 \mu\text{m}$. This value ensures that the boundary layer non dimensional parameter y^+ varies between 4 and 8 around the hydrofoil.

2.b) Influence of the turbulence model

Concerning the different turbulence models, it has to be noted that performing steady computations was not possible with all the models proposed in FINE/TURBO™. As a matter of fact, the following models didn't lead to convergence : $k-\epsilon$ Chien [11], Launder-Sharma [12], Yang-Shih and non-linear low Reynolds. Indeed, these models predicted a separation near the leading edge generating a periodic shedding of vorticity, which is incompatible with the hypothesis of converging a steady solution. This is illustrated by fig.3 where one can see the pressure coefficient field around the foil for an unsteady computation with Yang-Shih model. The regions of low pressure correspond to vortices.

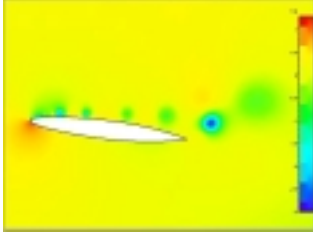


Figure 5: periodic shedding of vortices (instantaneous non-dimensional pressure field)

On the other hand, Baldwin-Lomax [13], Spalart-Allmaras [14] and k- ϵ with extended wall functions converged towards steady solutions.

The comparison between the different models is made on the basis of the predicted values of lift and drag coefficients, in which the viscous and pressure parts are analysed separately.

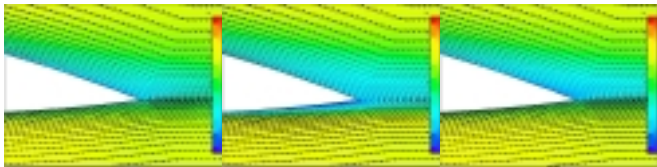
Models	Lift coefficient		
	Viscous	Pressure	Total
Baldwin-Lomax	-0.0006	0.6490	0.6484
Spalart-Allmaras	-0.0008	0.5642	0.5634
k- ϵ wall functions	-0.0007	0.6228	0.6221

Table 1: turbulence effect on lift

Models	Drag coefficient		
	Viscous	Pressure	Total
Baldwin-Lomax	0.0087	0.0210	0.0297
Spalart-Allmaras	0.0097	0.0204	0.0300
k- ϵ wall functions	0.0069	0.0225	0.0294

Table 2: turbulence effect on drag

The results show a good agreement on the predictions of total drag but it is very interesting to note that the sharing viscous / inviscid shows significant differences. Concerning the lift, a good agreement is obtained between Baldwin-Lomax and k- ϵ with extended wall functions. The Spalart-Allmaras model predicts a value that is 10% lower. The differences are due to the calculation of a separated flow near the trailing edge. The differences are illustrated on figure 6, where one can see the non dimensional pressure field and the velocity field:



a) Baldwin-Lomax b) k- ϵ wall function c) Spalart-Allmaras
Figure 6: boundary layer at trailing edge

Finally, we decided to use for cavitating calculations the k- ϵ model with extended wall functions. This choice was motivated by the possibility of studying the evolution of k and ϵ in the cavitating zone. However, the best choice would probably have been to use Spalart-Allmaras model. This could not be done as this model has only been implemented very recently by

Numeca International in its code (FINE/TURBO™ v6.1) and we did not have enough time to perform unsteady cavitating simulations.

3) CAVITATING ANALYSIS

This section presents results obtained for two different values of the cavitation parameter: $\sigma=0,8$ (characterised by an unsteady behaviour) and $\sigma=0,4$ (corresponding to a supercavitating sheet).

The following methodology has been applied to study the two configurations. First, a steady computation has been performed to obtain a global description. Then, an unsteady analysis has been continued from the base of the results given by the steady computation. This analysis begins by a research of the numerical conditions necessary to simulate the phenomena. Then, a discussion of the physical modelling aspects is performed and finally, the flow field computed is analysed.

3.a) Turbulence modelling

In k- ϵ models, the turbulent viscosity is computed as

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

As it had been previously shown from computations of cavitating flows in venturi type ducts by [15-3], this formulation doesn't allow the development of unsteadiness with the barotropic cavitation model. On the present test case, the same conclusion can be driven as it can be observed on the next figure.

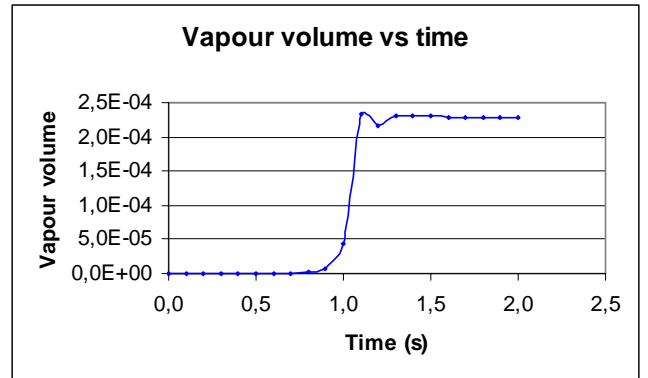


Figure 7: evolution of vapour volume

Therefore, we decided to use the modification presented in [15-3] to simulate an unsteady behaviour:

$$\mu_t = f(\rho) \rho C_\mu \frac{k^2}{\epsilon}$$

$$\text{where } f(\rho) = \frac{\rho_v + \alpha^n (\rho_l - \rho_v)}{\rho}$$

With a sufficiently high power n (n=10), a periodic shedding of vapour clouds is observed. This phenomenon will be studied in the section devoted to the analysis of the results.

3.b) Unsteady cavitation: $\sigma = 0,8$

To predict the behaviour of the flow for this value of σ , we first performed a steady computation that lead to the following results:

C_L	0.4633
C_D	0.0533
L_d	0
L_{max}	0,52
t_{max}	0,14
l_{tmax}	0,96
V	2,03e-04

Table 3: steady predictions

As the cavity was quite long, it seemed obvious that an unsteady behaviour should develop. The numerical parameters that have been used in the computation are given in table 4.

Δt	0,0002
CFL	1,5
NCAV	2000
RELAX	0,05
$\kappa^{(2)}$	1,0
ITMAX	100

Table 4: numerical parameters

- Analysis of the results

First, we will give a description of the flow through visualisations of the density field. This evolution will be compared to the variations of lift and drag and a mechanism will be proposed to the phenomenon observed.

- Description of the calculated flow

On figure 8, one can see the density field for several time steps. The visualisations present a cycle of cloud shedding. The liquid density is in red and as the lowest value of density is 200 kg/m^3 , this corresponds to dark blue.

Visualisations of the velocity field show the development of a re-entrant jet along the hydrofoil, which is in agreement with the classical theory explaining the periodic shedding of vapour structures downstream from a cavity [16].

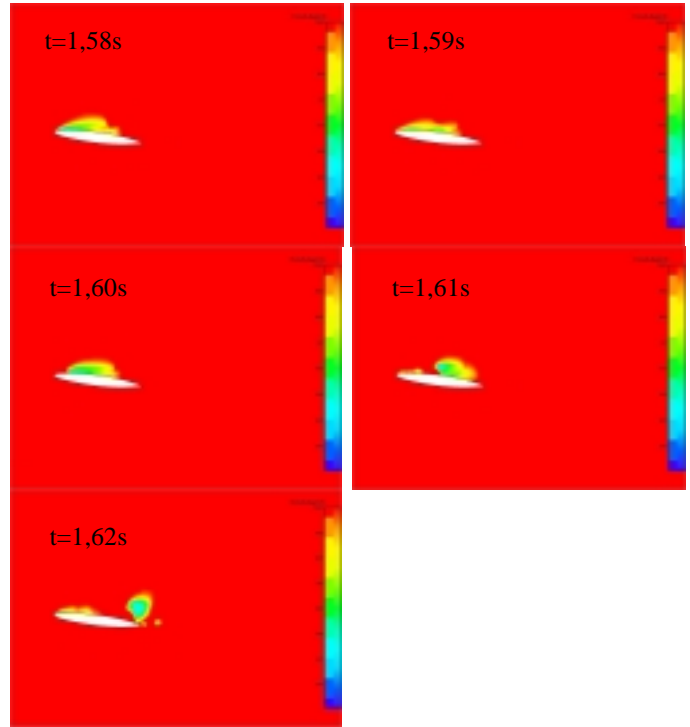
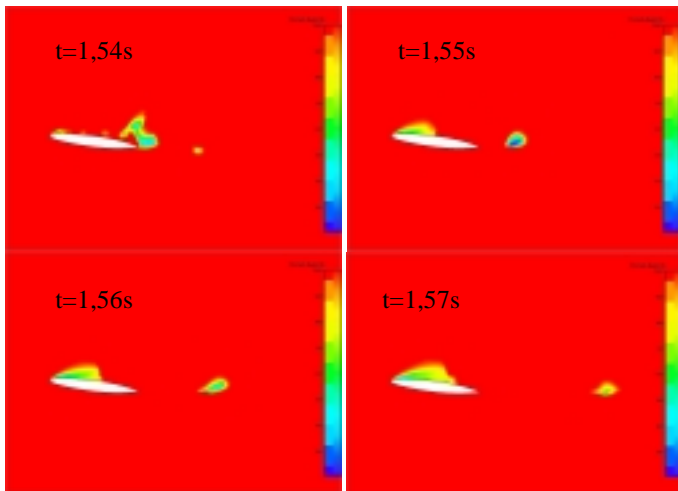


Figure 8: evolution of the density field

- Quantities of interest:

In this section, the averaged quantities of interest and some time dependent variations will be presented:

C_L	0,4566
C_D	0,0783
L_d	0
L_{max}	0,57
t_{max}	0,15
l_{tmax}	0,95
V	1,85e-04
St	0,091

Table 5: averaged quantities

The comparison of the steady and unsteady computations reveals that the main difference concerns the prediction of drag. The other quantities remain comparable.

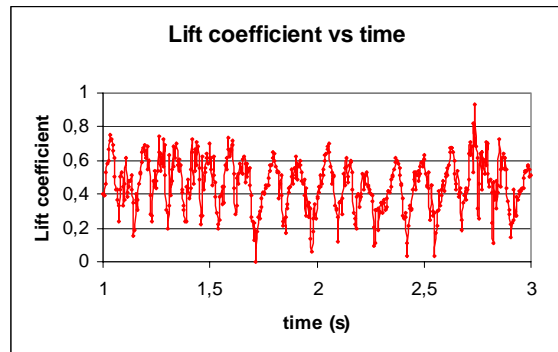


Figure 9: time evolution of lift coefficient

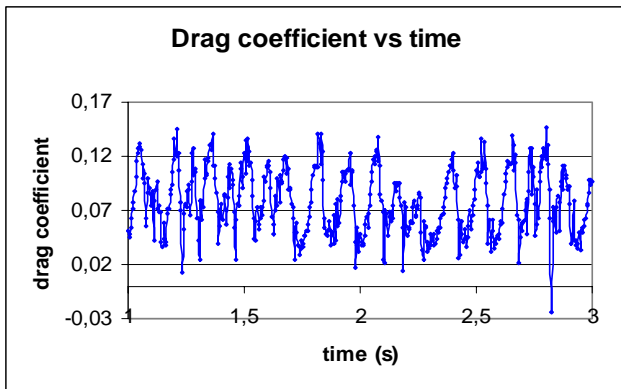


Figure 10: time evolution of drag coefficient

The time evolutions of lift and drag (fig. 9-10) are very similar. One can see that the signals are quite complicated but they underline the simultaneous development of phenomena at a low and at a higher frequency.

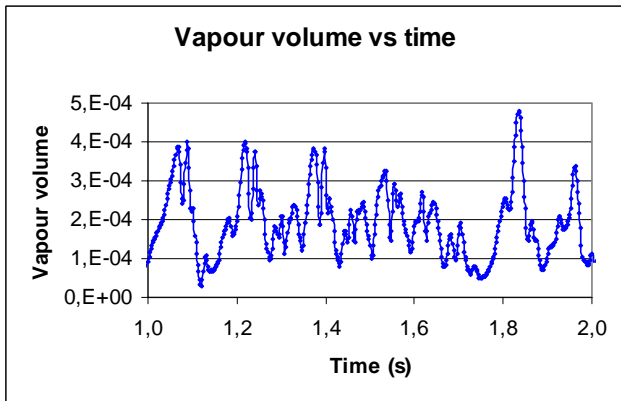


Figure 11: time evolution of vapour volume

The time evolution of total vapour volume is more chaotic than the evolution of lift and drag. This can be explained by the fact that vapour volume is integrated in the whole domain. Consequently, all the dynamics of the flow is mixed and local phenomena cannot be studied. On the contrary, lift and drag remain good indicators of the local changes of the flow in the vicinity of the foil and more particularly in the trailing edge area.

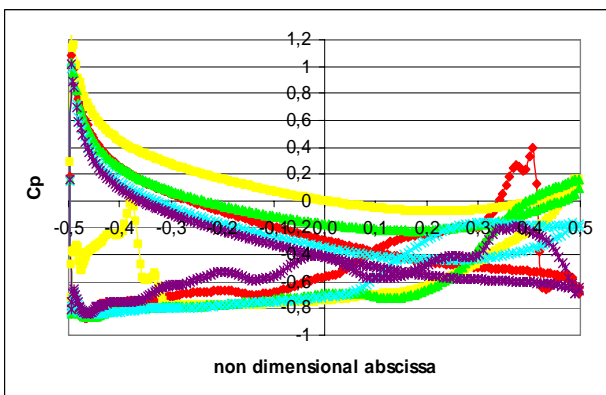


Figure 12: time evolution of Cp on the foil

The C_p curves are representative of the cycle illustrated on fig. 8. The first picture of fig.8 must be associated with the violet curve ($t=1,54s$), which corresponds to the beginning of the cycle (lift is minimum). The other curves are associated to the following times : light blue ($t=1,56s$), green ($t=1,58s$), yellow ($t=1,60s$) and red ($t=1,62s$).

o Physical mechanisms:

As it can be seen, the cycles are quite complicated and different frequencies appear to be significant. In this section, we will focus on two phenomena :

The evolution of the density field can be described as follows and associated to the lift evolution presented on figure 13. From the lowest value of lift, the vapour clouds are convected downstream from the foil and an attached cavity appears and grows from the leading edge. Then, this phase of growth stops and the cavity length decreases until the cavity is detached from the leading edge and convected along the foil. The analysis of the flow field proved that the attached cavity development was stopped when the cloud convected downstream from the foil collapses.

The evolution of lift shows two cycles. Lift was found first increasing, then decreasing and finally increasing again to reach a plateau. The cycle ends by a very abrupt drop.

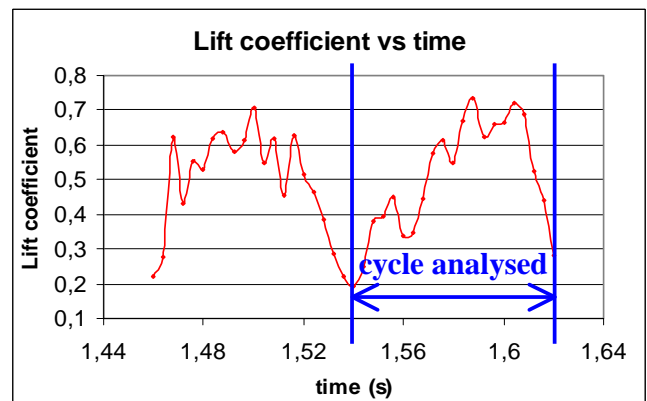


Figure 13: two cycles of lift evolution

The point of interest here is the first decrease. Why is the increasing trend stopped ? We first thought that the mechanism evoked previously was generating this artefact in the lift evolution but the comparison with the evolution of the density field showed us that the phenomena weren't synchronous. The explanation stands in the sudden increase of static pressure on the suction side that can be observed on fig. 13 (light blue curve, $t = 1.56s$). This modification of the static pressure is caused by the reattachment of the flow downstream from the cavity. Finally, it seems to us that the high frequency fluctuations can be associated to the modification of the flow on the trailing edge induced by the shedding of vorticity.

- Influence of the artificial dissipation

In the central scheme that we use to solve the Navier-Stokes equations an artificial dissipation [8] with 2nd and 4th order derivatives of the conservative variables has to be added in the calculation of the fluxes.

$$d_{i+\frac{1}{2}} = \varepsilon_{i+\frac{1}{2}}^{(2)} \delta U_{i+\frac{1}{2}} + \varepsilon_i^{(4)} \delta^3 U_{i+\frac{1}{2}}$$

The scalar coefficients ε are given by:

$$\varepsilon_{i+\frac{1}{2}}^{(2)} = \frac{1}{2} \kappa^{(2)} \lambda^* \max(v_{i-1}, v_i, v_{i+1}, v_{i+2})$$

$$\varepsilon_{i+\frac{1}{2}}^{(4)} = \max(0, \frac{1}{2} \kappa^{(4)} \lambda^* - \varepsilon_{i+\frac{1}{2}}^{(2)})$$

The parameter that is the object of the discussion is the coefficient $\kappa^{(2)}$ that multiplies the dissipation scaled on local pressure and density gradients. Indeed, in a study that we previously performed on a venturi type section [17], we observed that a very low value of $\kappa^{(2)}$ had to be imposed. For the present computation, the value used for the venturi was obviously too low to avoid the divergence. The analysis of the flow could show us that the source of numerical instability was the vapour travelling through the wake downstream from the foil. To run the present computation, a value of 1.0 was found to be necessary which is the classical value for turbomachinery flows but that is ten times the value required for the venturi computations. This seems important to us because it highlights the fact that a non dimensional criterion on the level of dissipation should be defined.

3.c) Supercavitation : $\sigma = 0,4$

For $\sigma = 0,4$ a steady supercavitating solution was simulated. In the present case, the unsteady computations gave some very different results. Consequently, we will first describe the steady solution and then present the unsteady analysis.

- Steady solution
 - Numerical aspects

A steady computation and an unsteady one with a time step $\Delta t=0,0005s$ were performed and gave results that were found very similar in that second case. The numerical parameters of the unsteady computation were the same than the one used for $\sigma=0,8$ (table 1). The stabilisation of this computation is illustrated by fig.14.

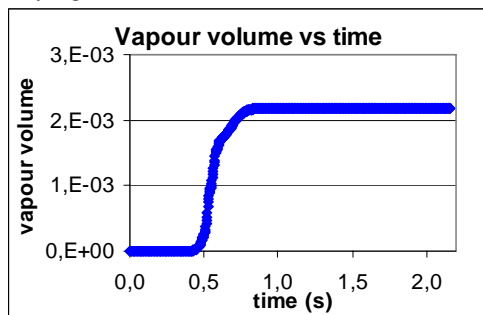


Figure 14: evolution of vapour volume

- Description of the flow

The next figure presents the density field around the foil.

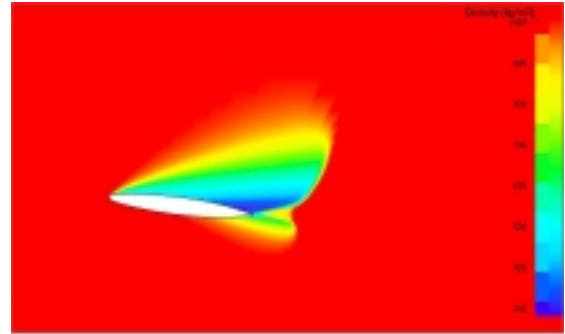


Figure 15: density field

It can be seen that the cavity develops downstream from the trailing edge and even on the pressure side. No re-entrant jet could be observed along the suction side.

C _l		
Viscous	Pressure	Total
-0.0006	0.2922	0.2916

C _d		
Viscous	Pressure	Total
0.0825	0.004	0.0865

Table 6: lift and drag predictions

The comparison of these values with those obtained for non cavitating conditions and for $\sigma=0,8$ shows that the lift has been strongly reduced. The value of drag was on the contrary found quite similar to the averaged value obtained with the unsteady computation at $\sigma=0,8$, but much higher than the values predicted with the steady computation, in non cavitating and partial cavitation conditions. The modifications are also significant on the sharing of the drag, now mainly due to viscous effects. Table 7 provides the values of the quantities chosen for the discussion on modelling.

C _L	0,2911
C _D	0,0866
L _d	0
L _{max}	1,5
t _{max}	0,5
l _{tmax}	1,00
V (m ³)	21,1e-04

Table 7: steady quantities

- Unsteady analysis

The time step used in the unsteady computation was based on the cavity length predicted by the steady computation. However, after many discussions on the physical relevance of such a solution, we decided to try lower values of the time step. With $\Delta t = 0,0002s$ (same value than for $\sigma = 0,8$) the solution seems to

become unsteady. Unfortunately, the computation is not finished yet. Consequently, we prefer not to provide partial results that would be insignificant in regards with the necessity of defining averaged values. On the other hand, those results may be available for the paper discussion.

CONCLUSION

The flow around the CAV2003 hydrofoil was investigated for three configurations. The non cavitating conditions were studied more particularly to compare different turbulence models. Three of them (Baldwin-Lomax, Spalart-Allmaras and $k-\epsilon$ with extended wall functions) allowed the simulation of a steady flow. The comparison of the predictions of lift and drag have highlighted the major role played by the flow in the area located at the immediate vicinity of the trailing edge.

For the first cavitating point ($\sigma = 0,8$), an unsteady cavitating behaviour was simulated. The computations performed have shown the importance of reducing the turbulent viscosity in two phase flows. Moreover, a comparison between a steady and an unsteady computation has shown that if lift, cavity dimensions and vapour volume were comparable, the value of drag was strongly under-predicted by the steady computation. Finally, the simulation of the second cavitating point ($\sigma = 0,4$) predicted a stable supercavitating behaviour for a time step of 0,0005s. Actually, a time step value of 0,0002s seems to allow the development of unsteadiness, which highlights the crucial importance of that parameter for the triggering of cavitation instabilities.

In the future, we have planed to continue this study with the aim to analyse more accurately the turbulence effects and the physical mechanisms that appear in the unsteady phenomena. In that frame, the comparison of our results with those obtained by other physical and numerical models may be very interesting.

ACKNOWLEDGMENTS

This research was supported by a doctoral grant from the French space agency CNES and SNECMA Moteurs DMS. The authors wish also to express their gratitude to NUMECA International for their technical support and more particularly to Eric Lorrain.

REFERENCES

- [1] O. Coutier-Delgosha, R. Fortes-Patella, J.-L. Reboud, B. Pouffary, « 3D Numerical Simulation Of Pump Cavitating Behaviour », Proceedings of ASME FEDSM'02, Montreal, Quebec, 2002.
- [2] Delannoy, Y., Kueny, J.L. (1990): "Two phase flow approach in unsteady cavitation modelling", Cavitation and Multiphase Flow Forum, ASME-FED vol.98, pp. 153-158.
- [3] Reboud J-L., Stutz B., Coutier O. (1998), "Two phase flow structure of cavitation: experiment and modeling of unsteady effects", Proceedings of the 3rd Int. Symp. on Cavitation, Grenoble, France.

[4] Merkle C.L., Feng J., Buelow P.E.O. (1998), "Computational modeling of the dynamics of sheet cavitation", 3rd Int. Symp. on Cavitation, Grenoble, France.

[5] Song and He (1998), "Numerical simulation of cavitating flows by single-phase flow approach", 3rd Int. Symp. on Cavitation, Grenoble, France.

[6] Kunz, R., Boger, D., Chyczewski, T., Stinebring, D., Gibeling, H. (1999), "Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies", 3rd ASME/JSME Joint Fluids Engineering Conference, San Francisco.

[7] Hakimi, N. (1997): "Preconditioning methods for time dependent Navier-Stokes equations", Ph.D.Thesis, Vrije Univ. Brussels.

[8] Jameson A. (1991), "Time dependant calculations using multigrid, with application to unsteady flows past airfoils and wings", AIAA Paper 91-1596.

[9] Turkel, E. (1987), "Preconditioning methods for solving the incompressible and low speed compressible equations", Journal of Comp. Phys., vol 72, pp. 277-298.

[10] Choi, D., Merkle, C. L. (1993), "The application of preconditioning in viscous flows", Journal of Comp. Phys., vol 105, pp. 207-223.

[11] Chien, K.Y. (1982), "Predictions of Channeland Boundary-Layer Flows with a Low-Reynolds Number Turbulence Model", AIAA J., vol 20, No. 1.

[12] Launder, B.E., Sharma, B. I. (1974), "Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow near a Spinning Disk", Letters in Heat and Mass Transfer, vol 1, pp. 131-138.

[13] Baldwin, B., Lomax, H. (1978), "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows", AIAA-78-257.

[14] Spalart, P.R., Allmaras, S.R. (1992), "A one equation turbulence model for aerodynamic flows", AIAA 92-0439.

[15] Coutier-Delgosha O., Fortes-Patella R. Reboud J-L., (2002), "Evaluation of the turbulence model influence on the numerical simulation of unsteady cavitation", J. Fluid Eng. Vol 25, pp. 38-45

[16] Callenaere, M., Franc, J. P., Michel, J.M., Riondet, M. (2001), "The cavitation instability induced by the development of a re-entrant jet", Journal of Fluid. Mec., vol 444, pp. 223-256.

[17] Pouffary, B., Lorrain, E. , Fortes-Patella R. Reboud J-L. (2003), "Numerical Simulation of Cavitating Flows in Space Pumps", Proceedings of the Workshop on Industrial Challenges in the Simulation of Evolving Interfaces, Sept.1-2, 20003, Vrije Universiteit Brussel, Brussels, Belgium.