NUMERICAL MODELING OF CAVITATING FLOWS AROUND HYDROFOILS

Carlos A. A. Carbonel H., ccarbonelh@unmsm.edu.pe
Universidad Nacional Mayor de San Marcos UNMSM, Av. Venezuela S/N, Lima, Perú

Abstract: This paper present a study of using a proposed finite element model to predict cavitating flow motions around two dimensional hydrofoils. The proposed model solves the governing equations of momentum and mass conservation including advection, pressure and shear stress terms. To describe turbulence influences around the hydrofoil the Prandtl-Kolgomorov model is included. The cavitating conditions are modeled through a mixture model involving liquid and vapor flows. The water vapor fraction is evaluated using a transport equation with source and sink for evaporation and condensation. The finite element model uses linear spatial polynomials to approximate the variables, a characteristic scheme to approach the non-linear advection terms and non-reflecting boundary conditions for the open outlet sides of the domain. Numerical experiments are performed for cavitation numbers 0.9 and 0.5 considering the NACA 0015 hydrofoil profile. The model is able to predict the essential features of cavitating flows and have the tendency to produce from medium to large cavities of vapor volume fraction directed to the downward side of the profile.

Keywords: Finite Element Model; Unsteady Flows; Mixture flows; Cavitation; Hydrofoils.

1. INTRODUCTION

The fluid motion around submersed bodies encompass a variety of fluid mechanics phenomena. The character of the flow field depends on the shape of the body. One of these phenomena is the Cavitation, which is a sequence of vaporization and condensation processes during high velocity of the fluid flow due the presence of a body. The mechanisms involved in the cavitation are not well understood and are a challenge for research and here an exploring study is conducted.

Fluid around bodies produces complex processes because the pattern and related forces depend strongly on various parameters such as size, orientation, speed and fluid properties. The resulting pressure and velocity field around the body is modified due the geometry and friction at the boundaries of the body. When the Reynolds number increases, the flow begins to separates with the formation of unsteady vortex motions mainly behind the body (Prandtl and Tietjens, 1957). The turbulent behavior of the fluid is an open question and exists different options to model the turbulence. The more used turbulent formulations are the family of $k-\epsilon$ models. A problem of these kind of models is that they are dependents of the geometry of the case considered and also suffers from the deficiencies of the gradient ansatz (Oertel, 2004). Due this fact, the presence of many constant is the characteristic of these models. In spite of this, it is frequently used in many software packages. Other option is the use of zero or one equation turbulent models (e.g. Smagorinsky model, Prandtl-Kolmogorov model).

Cavitation modeling is an open question, because numerical uncertainties and turbulence modeling produce significant differences in the results (Salvatore et al, 2009). Additionally, different turbulent models can lead to discrepancies in local and integral quantities. It is suspected that unsteadiness of cavitation isamped by viscosity. The deficiency of the standard models were also observed by different authors (Li et al, 2010). Therefore, the present investigation is based on a proposed finite element model with some particular features to reduce the numerical contamination due unrealistic advection approach and boundaries treatment. The influence of some parameters and the resulting volume vapor fraction will be studied numerically.

Many numerical solutions were reported in the literature about flow motions over hydrofoils. Mostly of them using finite volume techniques and finite differences (e.g. Mostafa et. al. 2010; Karim et. al, 2010;
Kawamura and Sakoda, 2003; etc). Finite element approximations are still for the most part unexplored. The present paper is a step in the study of such problems using finite elements which is a powerful numerical technique to solve engineering problems (Hughes T., 2000; Connor and Brebbia, 1980).

2. THE HYDRODYNAMIC MODEL

2.1 Governing equations

The hydrodynamic incompressible flow around a body in two dimensions are described by the momentum and mass equations of an Newtonian fluid of density $\rho$ in a vertical cartesian coordinate system $(x_1, x_2)$ with turbulent velocities $\bar{u} = (u_1, u_2)$, pressure $p$, $\bar{g} = (0, g)$ and eddy viscosity $\nu_T$:

$$\rho \left( \frac{\partial \bar{u}}{\partial t} + \bar{u} \nabla \bar{u} \right) + \nabla p - \rho \nu_T \Delta \bar{u} - \rho \bar{g} = 0,$$

$$\nabla \bar{u} = 0,$$

where the two-dimensional vector operator $\left( \frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right)$ is denoted by $\nabla$ and the divergence $\Delta = \nabla \nabla$.

Non slip boundary conditions are prescribed on the surface of submersed body $\Gamma_H$,

$$u = 0 \quad \text{on} \quad \Gamma_H,$$

essential boundary condition on the entrance boundary $\Gamma_E$,

$$u = u_\infty \quad \text{on} \quad \Gamma_E,$$

natural boundary conditions on the boundary tunnel walls $\Gamma_C$,

$$\frac{\partial \bar{u}}{\partial n} = 0 \quad \text{on} \quad \Gamma_C,$$

and weakly reflective conditions on the outlet side boundary $\Gamma_O$ of the flow domain

$$\rho \frac{\partial \bar{u}}{\partial t} + \nabla_n p = 0 \quad \text{on} \quad \Gamma_O.$$

The eddy viscosity is modeled following the Prandtl-Kolmogorov turbulent model using

$$\nu_t = c \rho l \sqrt{k},$$

where $c$ is a constant equal to 0.54, $k$ is the turbulent kinetic energy, $l$ is the characteristic mixing length, $\rho$ is the fluid density. To evaluate $\nu_T$ the kinetic energy $k$ is modeled using:

$$\frac{\partial k}{\partial t} + \bar{u} \nabla k - \nabla \frac{1}{\sigma} \nu_T \nabla k - \nu_T |\nabla \bar{u} + \nabla \bar{u}^T|^2 + \varepsilon = 0,$$

where $\varepsilon$ is the dissipation of kinetic energy approached as

$$\varepsilon = \rho c \varepsilon k^{3/2} l.$$

Cases of mixture flows could be studied considering a simple single fluid approach and a transport equation for the vapor volume fraction. The transport equation describes the mixture model proposed by Singhal et. al. (2002) which is written as

$$\frac{\partial \rho f}{\partial t} + \nabla \rho f \bar{u} - (S_e - S_v) = 0$$

where $\rho$ is the mixture density, $f$ is the vapour mass fraction and

$$S_e = C_e \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \sqrt{\frac{2P_v - P}{\rho_l}} (1 - f) \quad P < P_v$$

$$S_e = C_e \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \sqrt{\frac{2P - P_v}{\rho_l}} f \quad P > P_v$$
\(S_e, S_c\) represent the source terms for vapor generation and vapor condensation respectively.

The relation between the density mixture \(\rho\) and the vapour mass fraction \(f\) is described by

\[
\frac{1}{\rho} = \frac{f}{\rho_v} + \frac{1-f}{\rho_l},
\]

and the volume fraction of vapour phase \(\alpha_v\) is related to the vapour mass fraction \(f\) according to:

\[
\alpha_v = \frac{f}{\rho} \frac{\rho_v}{\rho_l}.
\]

### 2.2 The variational weak formulation

The variational weak formulation of the unsteady hydrodynamic boundary value problem reads: Find \([\bar{u}, p, k, f]\) in a suitable functional \(S\) such that for a set of admissible test functions \([w^n, w^p, w^k, w^f] \in V\) satisfy:

\[
\int_{\Omega} w^n \left( \rho \left( \frac{\partial \bar{u}}{\partial t} + \bar{u} \nabla \bar{u} \right) + \nabla p + \rho \nu \Delta \bar{u} - \rho g \right) \, d\Omega = 0,
\]

\[
\int_{\Omega} w^p \nabla \bar{u} \cdot d\Omega = 0,
\]

\[
\int_{\Omega} w^k \left( \frac{\partial \bar{k}}{\partial t} + \bar{u} \nabla \bar{k} - \nabla \frac{1}{\sigma} \nu \nabla \bar{k} - \nu \nabla |\nabla \bar{u} + \nabla \bar{u}^T|^2 + \epsilon \right) \, d\Omega = 0,
\]

\[
\int_{\Omega} w^f \left( \frac{\partial f}{\partial t} + \nabla f \rho \bar{u} - S_e + S_c \right) \, d\Omega = 0,
\]

and also the boundary constraints

\[
\int_{\Gamma_O} w^n \left( \rho \frac{\partial \bar{u}}{\partial n} + \nabla n p \right) \, d\Gamma = 0 \quad \text{on} \quad \Gamma_O,
\]

where the subscript “\(n\)" indicates the normal component of the velocity component and gradient.

The two-dimensional spacial domain \(\Omega\) is partitioned in \(N_{\text{el}}\) triangular subdomains \(\Omega_e\) with a resulting number of nodes \(N_{\text{nod}}\). Similarly, the time domain is also paritioned into subintervals \(T^n = [t^n, t^{n+1}]\) of length \(\Delta t\), where the time levels belong to an ordered partition

\[
0 = t^0 < t^1 < t^2 < \ldots \ldots < t^M = T,
\]

where \(T\) is the end time.

The discretization proceeds by introducing the finite element expansion given by

\[
u_1 = u_1(t) \phi_j, \quad u_2 = u_2(t) \phi_j, \quad p = p_j(t) \phi_j, \quad k = k_j(t) \phi_j, \quad f = f_j(t) \phi_j \quad \text{with} \quad j = 1, 2, \ldots, N_{\text{nod}}
\]

such that \(\bar{u}_j(t) = [u_1(t), u_2(t)]\) and \(\phi_j(x_1, x_2)\) are the linear basis functions.

Substituting the variables by their approaches (21), the variational formulation of the indicated equations after integration by parts is written in the following global form

\[
M^n \frac{d \bar{u}_j(t)}{dt} + H^n p_j(t) - R^n u_j(t) - G_u = 0,
\]

\[
H^n \bar{u}_j(t) = 0,
\]

\[
M^k \frac{d k_j(t)}{dt} - R^k k_j(t) - P^k + D^k = 0,
\]

\[
M^f \frac{d f_j(t)}{dt} - S^f = 0,
\]

for \(j = 1, 2, \ldots, N_{\text{nod}}\).
The matrices $M^u$, $M^k$, $M^f$, $H^u$, $H^c$, $R^u$, $R^k$ and vectors $G_u$, $P^k$, $D^k$, $S^f$ represent the terms composed by the integrals considered in the variational formulations

$M^u = \int_\Omega w^u \rho (\phi_j, \phi_j) \, d\Omega,$
$H^u = \int_\Omega w^u \nabla (\phi_j, \phi_j) \, d\Omega,$
$R^u = \int_\Omega \nabla w \rho \bar{\phi}_j \nabla (\phi_j, \phi_j) \, d\Omega,$

(26)

$G^u = \int_\Omega w^u \rho \bar{\phi} \, d\Omega,$
$H^c = \int_\Omega w^c \nabla (\phi_j, \phi_j) \, d\Omega,$

(27)

$M^k = \int_\Omega w^k \phi_j \, d\Omega,$
$R^k = \int_\Omega \nabla w^k \frac{1}{\sigma} \nu T \phi_j \, d\Omega,$
$P^k = \int_\Omega w^k \nu T |\nabla \bar{u} + \nabla \bar{u} T|^2 \, d\Omega,$

(28)

$D^k = \int_\Omega w^k \phi_j \, d\Omega,$
$M^f = \int_\Omega w^f \rho \phi_j \, d\Omega,$

(29)

$M^f = \int_\Omega w^f \phi_j \, d\Omega,$
$S^f = \int_\Omega w^f (S_c - \bar{S}_c) \, d\Omega.$

(30)

In the present formulation the test functions $[w^u, w^c, w^k, w^f]$ are setting equal to $\phi_j$. The approximation of time derivatives are obtained defining for a generic variable $U(t)$ a linear approach between the two time levels $n$ and $n+1$ expressed as

$U(t) = \theta U^{n+1} + (1 - \theta) U^n,$

(31)

where

$\theta = \frac{t - t^n}{t^{n+1} - t^n}.$

(32)

In this way the time derivative is approached by

$\frac{d}{dt} U(t) = \frac{1}{\Delta t} (U^{n+1} - U^n),$

(33)

in the present paper $\theta$ was fixed equal 1.

The numerical integration in time has an implicit form of the equation as follows

$M^u \left( \frac{\bar{u}_j^{n+1} - \bar{u}_j^n}{\Delta t} \right) + H^u p_j^{n+1} - R^u \bar{u}_j^{n+1} - G_u = 0,$

(34)

$H^c \bar{u}_j^{n+1} = 0,$

(35)

$M^k \left( \frac{k_j^{n+1} - k_j^n}{\Delta t} \right) - R^k k_j^{n+1} - P^k + D^k = 0,$

(36)

$M^f \left( \frac{f_j^{n+1} - f_j^n}{\Delta t} \right) - S^f = 0.$

(37)

The terms $\bar{u}_j^n, \bar{p}_j^n, k_j^n, f_j^n$ are the variables at time level “$n$” obtained by a characteristic approach. This is a way to calculate these values when nonlinear advection take place. The characteristics are based on the fact that motions invariants propagates along the characteristics in space and time conserving a Riemann Invariant (Abbot, 1966). Also mixing the method of characteristics and the finite element method (Pironneau, 1982) gives satisfactory solutions. A generic variable $\bar{U}_j^n$ is function of a vector field $V$ and the particle path $X(x_1, x_2)$, such that $X$ is solved integrating backward in time (to get $X$) the following equation:

$\frac{dX}{dt} = V(X, t)$

when $X(\Delta t(n+1)) = (x_1, x_2).$

(38)
3. NUMERICAL EXPERIMENTS

3.1 Basic parameters of the experiments

The experiments conducted in this paper are based on the flow field over a NACA 0015 symmetric hydrofoil submersed in water. The hydrofoil of chord length \( c = 0.1 \text{m} \) is studied numerically in the present section. A tunnel of length 10\( c \) and height 4\( c \) is considered. The hydrofoil is located in the middle of the channel (Figure 1). The finite element mesh is composed by triangular elements. The number of triangular elements are 14120 with 7220 nodes. The reference pressure \( p_{\infty} \) increases with the depth. Slip boundary conditions are imposed in the upper and lower tunnel walls. Non-slip conditions are imposed on the surface of the hydrofoil. In the outlet boundary a suitable boundary condition is imposed, whereas in the entrance boundary a uniform velocity \( u_{\infty} \) is imposed.

![Figure 1. Hydrodynamic channel mesh](image)

This NACA 0015 profile is often used in aero and hydrodynamic studies. Here, the unsteady hydrodynamic characteristics of the hydrofoil is explored during cavitating conditions. The parameter used in the experiments are the following: the water density at 25\(^0\)C is fixed as \( \rho_l = 997.00900 \text{ kgm}^{-3} \), and the dynamic viscosity is equal to \( \mu_l = 8.91 \times 10^{-4} \text{ Pas} \). When the experiment are related to cavitating conditions, the vapour water density is \( \rho_v = 0.02308 \text{ kgm}^{-3} \) and the vapour dynamic viscosity is \( \mu_v = 9.8626 \times 10^{-6} \text{ Pas} \) and the vaporisation pressure is \( P_v = 3169 \text{ Pa} \). The integration in time used a time interval of \( \Delta t = 0.0005 \text{ s} \) up to time 0.5s.

For the \( C_p \) field, the comparison criteria is the pressure coefficient at the stagnation point is maximal and equal to the value \( C_p = 1 \). The pressure coefficient is defined as

\[
C_p = \frac{p - p_{\infty}}{\frac{1}{2}\rho u_{\infty}^2}.
\] (39)

In the present paper, the cavitation number \( \sigma \) describes the state conditions related to the saturation pressure \( p_v \) and is defined as

\[
\sigma = \frac{p_v - p}{\frac{1}{2}\rho u_{\infty}^2}.
\] (40)

The main interest of the experiments is mainly focused on the unsteady pattern of the vapor volume for some different parameter conditions.

3.2 Main solutions

The main solutions are performed for cavitation numbers \( \sigma = 0.9 \) and \( \sigma = 0.5 \) when \( l = c/1000 \). In these solutions the control for noncondensable vapor water mass is included in the experiments. It means that condensation is possible only when \( \alpha_v \) is greater than 0.3. The figures 2 and 3 show the vapor volume \( \alpha_v \) for \( \sigma = 0.9 \) and \( \sigma = 0.5 \) at different time instants. Also the \( C_p \) field at time 0.4s is presented in each case showing that negative \( C_p \) values are detected around the profile during the cavitation. The model solutions predict a large and wide cavity of vapor which survives as far downstream, a feature reported in observational experiments (Li et al, 2010).
3.3 Influence of elimination of noncondensable vapor mass

In the present experiment the calculation for $\sigma = 0.5$ is repeated but without to consider a vapor volume fraction of noncondensable flow. The Figure 4 shows different instant of the unsteady solution for the saturated vapor field $\alpha_v$. The $\alpha_v$ field shows a medium to large cloud cavities (trailing edge) due cavitating vortices downstream and its interference with the external flow.
3.4 Saturation pressure fluctuations

Effects of turbulence on cavitating flows has been reported in several experimental investigations. An approach of the turbulent influence is possible assuming a fluctuating pressure equal to $0.39 \rho k$. In the present experiment, the time dependent saturation pressure is approached by $p_v = 3169 + 0.39 \rho k/2$. 

Figure 4. Contours of vapor volume fraction $\alpha_v$ (without control of non-condensable mass) when $\sigma = 0.5$, at instants a)0.4s; b)0.45s and c)0.5s

Figure 5. Contours of vapor volume fraction $\alpha_v$ when the saturation pressure fluctuates due turbulence. $\sigma = 0.9$, at instants a)0.4s; b)0.45s and c)0.5s
The distributions of the vapor volume fraction when the saturation pressure fluctuations take place are presented in Figure 5. The distributions of $\alpha_v$ fields are similar to the results obtained in the main solution (Figure 2).

4. SUMMARY AND CONCLUSIONS

A two-dimensional finite element model is proposed to study cavitating flow motions around hydrofoils. The model solves the governing equations of momentum and mass conservation including advection, pressure and shear stress terms. The Prandtl-Kolgomorov turbulent model is included and to describe phase flows motions (liquid and vapor flows) a transport equation is used to simulate the evolution of the mixture of water mass vapor fraction. The finite element model includes a characteristic scheme to approach the non-linear advection terms in the equations and for the open outlet sides of the domain non-reflecting open boundary conditions are imposed.

The numerical experiments were performed to study the influence of different factors on the cavitating behavior around the NACA 0015 hydrofoil.

The experiments show the generation of vapor volume fraction in the upper side of the hydrofoil due the reduction of the water density forming a vapor cavity with unsteady behavior.

The main solutions consider no condensation for $\alpha_v \leq 0.3$. The main solutions ($\sigma = 0.5$ and $\sigma = 0.9$) are non-permanent solutions with low pressures located on the upper side, covering also partially the lower side of the profile. The model predicts large and wide vapor cavity. The intensity of the vapor volume generation is stronger when $\sigma = 0.5$ compared to the solutions with $\sigma = 0.9$.

When the condensation limit (no condensation for $\alpha_v \leq 0.3$) is avoided, the vapor volume is reduced but showing a medium and a large vapor cavity. A non-permanent behavior is better observed.

The fluctuations of saturation pressure does not modified in a significant way the cavity pattern of the vapor volume field.

REFERENCES


RESPONSABILITY NOTICE

The author is the only responsible for the printed material included in this paper.