NUMERICAL SIMULATION OF TWO-PHASE CAVITATING FLOW IN TURBOMACHINES

Sandor BERNAD, Senior Researcher*
Center of Advanced Research in Engineering Sciences
Romanian Academy - Timisoara Branch

Sebastian MUNTEAN, Senior Researcher
Center of Advanced Research in Engineering Sciences
Romanian Academy - Timisoara Branch

Romeo F. SUSAN-RESIGA, Professor
Department of Hydraulic Machinery
“Politehnica” University of Timisoara

Ioan ANTON, Prof., Member of Romanian Academy
Department of Hydraulic Machinery
“Politehnica” University of Timisoara

*Corresponding author: Bv Mihai Viteazu 24, 300223, Timisoara, Romania
Tel.: (+40) 256 403692, Fax: (+40) 256 403700, Email: sbernad@mh.mec.utt.ro

ABSTRACT

The paper briefly reviews the methodology currently employed for industrial cavitating flow simulations using the two-phase mixture model. It is shown how further theoretical developments can be included in expert commercial codes such as FLUENT using the User Defined Function facility.

The two-phase mixture model is evaluated and validated using two benchmark problems where experimental data are available. The first case is the axisymmetric hemispherical cavitating body, and the second one corresponds to a truncated NACA0009 hydrofoil. Both problems are solved using 2D (axisymmetric and plane, respectively) models.

Next, a 3D cavitating flow computation is performed for the GAMM Francis runner. The model is able to qualitatively predict the location and extent of the 3D cavity on the blade, but further investigation are needed to quantitatively assess the accuracy for real turbomachinery cavitating flows.

KEY WORDS

Cavitating flow numerical simulation, two-phase mixture cavitation model, hydraulic Francis turbine

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>[-]</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>$g$</td>
<td>[m/s$^2$]</td>
<td>gravity</td>
</tr>
<tr>
<td>$m$</td>
<td>[kg/(s.m$^3$)]</td>
<td>inter-phase mass flow rate per unit volume</td>
</tr>
<tr>
<td>$n_b$</td>
<td>[1/m$^3$]</td>
<td>number of bubbles per unit volume of liquid</td>
</tr>
<tr>
<td>$p$</td>
<td>[Pa]</td>
<td>pressure</td>
</tr>
<tr>
<td>$r$</td>
<td>[m]</td>
<td>radius, radial coordinate</td>
</tr>
<tr>
<td>$t$</td>
<td>[s]</td>
<td>time</td>
</tr>
<tr>
<td>$u$</td>
<td>[m/s]</td>
<td>absolute velocity</td>
</tr>
<tr>
<td>$x$</td>
<td>[m]</td>
<td>axial coordinate</td>
</tr>
<tr>
<td>$E$</td>
<td>[J/kg]</td>
<td>specific energy (energy per unit mass)</td>
</tr>
<tr>
<td>$H_s$</td>
<td>[m]</td>
<td>suction head</td>
</tr>
<tr>
<td>$R$</td>
<td>[m]</td>
<td>bubble radius</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>[-]</td>
<td>vapour volume fraction</td>
</tr>
<tr>
<td>$\rho$</td>
<td>[kg/m$^3$]</td>
<td>density</td>
</tr>
<tr>
<td>$\omega$</td>
<td>[1/s]</td>
<td>vorticity</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>[-]</td>
<td>cavitation number</td>
</tr>
</tbody>
</table>

Subscripts and Superscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m$</td>
<td>mixture</td>
</tr>
<tr>
<td>$v$</td>
<td>vapour of vaporization</td>
</tr>
<tr>
<td>$l$</td>
<td>liquid</td>
</tr>
<tr>
<td>$\infty$</td>
<td>points of large distance from the body</td>
</tr>
<tr>
<td>ref</td>
<td>reference point</td>
</tr>
<tr>
<td>atm</td>
<td>atmospheric conditions</td>
</tr>
<tr>
<td>PS, SS</td>
<td>pressure side, suction side</td>
</tr>
</tbody>
</table>

1. INTRODUCTION

Cavitation is one of the major problems that hinders the hydraulic machinery performances. Once the flow velocity and blade loading are increased there are regions where the pressure drops well below the vaporization pressure of the liquid and cavitation is developed. The associated noise and vibrations, as well as cavitation erosion, have motivated a great number of theoretical and experimental studies with
the aimed at elucidating both cavitation physics and its practical consequences such as erosion and vibrations [1]. So far, the cavitation bubble dynamics in Newtonian fluids and the associated phenomena are well understood. On the other hand, cavitation in hydraulic machinery is a complex, three-dimensional unsteady phenomenon which has yet to be studied both theoretically and experimentally. A great body of experimental data is currently available, [1], for cavitation flows in hydraulic machinery (turbines and pumps), as well as various hydro-mechanical configurations. Useful empirical correlations have been established in order to predict the cavitation behaviour from classical liquid flow computations. For example, cavitation inception can be well predicted. On the other hand, developed cavitation is more difficult to compute, since the cavity presence significantly changes the pressure field in its neighbourhood. Moreover, the single bubble dynamics can only offer a qualitative image, since large bubble clouds and/or cavities are usually present in hydraulic machines and equipment. The present paper is part of an ongoing effort to develop numerical techniques for cavitation modelling and applications to hydraulic turbines at the Hydrodynamics and Cavitation Laboratory from the Romanian Academy – Timișoara Branch [2, 16, 17].

Section 2 presents the homogeneous flow model for liquid-vapour cavitation flows. We use a transport equation for the vapour volume fraction, in addition to the continuity and momentum equations for the liquid-vapour mixture. The main issue here is to model the interphase mass flow rate. In this paper we are using a simplified model based on the Rayleigh equation that governs the cavitation bubble dynamics. This model is implemented in the commercial code FLUENT using the User Defined Function presented in the Appendix. The main advantage of this approach is that one can fully take advantage of the whole software infrastructure of an well established commercial code while using various cavitation models specifically designed for particular problems. Recent developments of this approach are also reviewed.

In Section 3 we validate the cavitation model by computing the cavitation flow over and axi-symmetric hemispherical fore-body. The correct physics of the problem (especially the cavity closure and the re-entrant jet) is recovered. Moreover, the computed pressure distribution is in excellent agreement with available experimental data. The same conclusions are obtained from the numerical investigation of an isolated hydrofoil cavitation flow.

Section 4 is devoted to the investigation of cavitation flow in the GAMM Francis turbine runner. The 3D cavity shape and extent is in good agreement with flow visualization data. We are also investigating the influence of the cavity on the local pressure distribution. For the case investigated in this paper, the cavity has little influence in a blade section (S15) near the runner band. Since we have an early cavitation stage, the influence on the runner torque is small. Actually, a small runner torque increase is obtained, in agreement with experiments in the industrially tolerated cavitation regime.

The conclusions are summarized in the last section, and the source code for the User Defined Function developed for the FLUENT code is presented in the Appendix.

2. CAVITATING FLOW MODELING

Numerical simulation of two-phase cavitation flows is an ongoing research effort with the ambitious goal to compute the unsteady evolution for cavities grow and collapse. The CFD community has developed so far a set of mature techniques for simulating single-phase viscous flows, and the past half century of accumulated experience may very well serve to shape the numerical cavitation flow research. Early studies rely on the potential flow theory [11]. This approach is now able to correctly describe partially cavitation two-dimensional hydrofoils, including the re-entrant jet cavity closure model [7]. However, extension to 3D problems and other types of cavitation flows seems to be out of reach for the potential flow model.

Although basic cavitation theoretical studies deal with bubble (or bubble clouds) dynamics by solving for the vapour-liquid interface, most of the practical cavitation flows are approached using a homogeneous flow theory. The main idea is to consider a single variable density fluid, without explicit phase interfaces. This model has emerged after carefully examining available experimental investigations, as well as by evaluating the computational costs involved in cavitation flows modelling. A review of cavitation flows numerical studies over the past decade can be found in [17], where various Reynolds Averaged Navier-Stokes (RANS) solvers have been modified to account for the secondary phase (vapour and gas) dynamics.

Let us briefly review here the mixture model. The mixture is a hypothetical fluid with variable density, $ho_m = \alpha \rho_v + (1 - \alpha) \rho_l$ (1)

Ranging from liquid density for $\alpha = 0$ to vapour density $\rho_v$ for $\alpha = 1$. The vapour volume fraction

$\alpha = \frac{Vol_{vapor}}{Vol_{liquid} + Vol_{vapor}}$ (2)

is an additional unknown of the problem. The mixture will of course satisfy the continuity equation.
\[ \frac{d \rho_m}{dt} + \rho_m \nabla \cdot \mathbf{u}_m = 0 \quad (3) \]

where \( \frac{d}{dt} \) denotes the material derivative. Next, one has to consider a momentum equation for the mixture. A simple choice would be to neglect the viscous effects and use the Euler equation. The system of equations can be then closed with a relationship density-pressure (equation of state). This approach can take advantage of a rich legacy of inviscid compressible solvers [6]. However, when considering a barotropic mixture, i.e. the density depends solely on the pressure, some physics is lost. This can be easily seen when writing the vorticity transport equation

\[ \frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \omega \cdot \nabla \mathbf{u} + \frac{1}{\rho^2} \nabla \rho \times \nabla p + \text{viscous terms} \quad (4) \]

The second term in the right-hand-side, which accounts for the baroclinic vorticity generation, vanishes when \( \rho = \rho(p) \). As a result, an important vorticity source is lost, especially in the cavity closure region [13].

Practical computations of industrial flows are using RANS equations with various turbulence modeling capabilities. This approach is embedded in most commercial codes currently available, e.g. FLUENT [18]. As a result, it seems natural to build a cavitating flow model on top of such computational infrastructure.

An alternative to the equation of state is to derive a transport equation for the vapour volume fraction. The continuity Eq. (3), together with Eq. (1), give the velocity divergence as

\[ \nabla \cdot \mathbf{u}_m = -\frac{1}{\rho_m} \frac{d \rho_m}{dt} = \frac{\rho_l - \rho_v}{\rho_m} \frac{d \alpha}{dt} \quad (5) \]

Using Eq. (5), the conservative form of the transport equation for \( \alpha \) can be easily written,

\[ \frac{\partial \alpha}{\partial t} + \mathbf{u} \cdot \nabla [(\alpha) \mathbf{u}_m] = \frac{1}{\rho_v} \left[ \frac{\rho_v}{\rho_m} \frac{d \alpha}{dt} \right] \quad (6) \]

Eq. (6), can be also written for the liquid volume fraction, \( 1 - \alpha \),

\[ \frac{\partial (1 - \alpha)}{\partial t} + \mathbf{u} \cdot \nabla [(1 - \alpha) \mathbf{u}_m] = \frac{1}{\rho_l} \left[ \frac{\rho_l}{\rho_m} \frac{d \alpha}{dt} \right] \quad (7) \]

The factor in square brackets in the r.h.s. of Eqs. (6) and (7) is the interphase mass flow rate per unit volume:

\[ \dot{m} = \frac{\rho_v}{\rho_m} \frac{d \alpha}{dt} \quad (8) \]

If we add Eqs. (6) and (7), we end up with an inhomogeneous continuity equation of the form

\[ \nabla \cdot \mathbf{u}_m = \dot{m} \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right) \quad (9) \]

which is used in [8] to replace homogeneous Eq (3). Finally, the vapour volume fraction transport equation is written as:

\[ \frac{\partial \alpha}{\partial t} + \mathbf{u} \cdot (\alpha \mathbf{u}_m) = \frac{1}{\rho_v} \dot{m} \quad (10) \]

This is the equation for the additional variable \( \alpha \), to be solved together with the continuity and momentum equations.

Most of the efforts in cavitation modelling are focused on correctly evaluating \( \dot{m} \).

One approach has been proposed by Merkle et al. [9], by modelling the phase transition process similar to the chemically reacting flows.

This model was successfully employed by Kunz et al. [8] in a variety of cavitating flows. However, the model constants are chosen somehow arbitrary, and this choice ranges several orders of magnitude from one paper to another. Senocak and Shyy attempt a derivation of an empiricism-free cavitation model [13] in order to avoid the evaporation/condensation parameters introduced by Merkle.

A different approach is proposed by Schnerr and Sauer [15], who consider the vapour-liquid mixture as containing a large number of spherical bubbles. As a result, the vapour volume fraction can be written as

\[ \alpha = \frac{n_b}{1 + n_b} \frac{4 \pi R^3}{3} \quad (11) \]

where the number of bubbles per volume of liquid, \( n_b \), is a parameter of the model. From (11) we can easily get

\[ \frac{d \alpha}{dt} = \alpha (1 - \alpha) \frac{3 \dot{R}}{R} \quad (12) \]

where \( \dot{R} \) is the bubble vapour-liquid interface velocity. A simplified Rayleigh equation can be used to compute

\[ \dot{R} = \frac{dR}{dt} = \text{sgn}(\rho_v - p) \sqrt{\frac{2 |p_v - p|}{\rho_l}} \quad (13) \]

Of course the bubble grows if the mixture pressure is less than the vaporization pressure, \( p < p_v \), and collapse when \( p > p_v \). The bubble collapse, as modelled by the Rayleigh second order differential equation, is much more rapid than the bubble growth. However, the above model seems to make no such difference between grow and collapse.
In a recent development, Singhal et al. [14] propose to account for the local turbulence intensity by correcting the vapor pressure as

\[ p_v = p_{\text{sat}} + \frac{p_{\text{turb}}}{2} \]

where \( p_{\text{sat}} \) is the saturation pressure corresponding to the liquid temperature, and \( p_{\text{turb}} = 0.39 \rho k \) accounts for the turbulent kinetic energy \( k \). This modification simply raises the phase-change threshold, and in practice it performs better near the cavity closure region where intense vorticity generation enhances the turbulence.

The present paper employs the mixture model, as implemented in the FLUENT commercial code, with the cavitation model described by Eqs. (8), (12) and (13) implemented through an User Defined Function (Appendix).

3. TWO-DIMENSIONAL CAVITATING FLOW COMPUTATION AND VALIDATION

Before any attempt of computing 3D cavitating turbomachinery flows, we have tested the model described in Section 2 on two benchmark problems.

First, the flow with and without cavitation computed for the axi-symmetric cavitator with hemispherical fore-body and numerical results are compared with experimental data of Rouse and McNown [12]. Figure 1 shows the distribution of \( \alpha \) around a hemispherical fore-body, for a cavitation number

\[ \sigma = \frac{p_{\infty} - p_{v}}{\frac{1}{2} \rho U_{\infty}^2} = 0.3 \]

Most of the computational domain contains only liquid, \( \alpha = 0 \), but within the region with \( p < p_{v} \) the vapour phase is formed with \( 0 < \alpha < 1 \).

Two particular streamlines originating from upstream the body, are displayed in Fig. 1. The streamline which is closer to the axis goes around the cavity then re-enters the cavity and ends up in a recirculating region. This streamline is embedded in the re-entrant jet which is known to be formed at the cavity closer. The second streamlines is very close to the first one around the cavity, once the cavity passed it goes further downstream along the body surface. The point where the two streamlines split is a downstream stagnation point. Within the cavity there are regions practically filled with gas (the first half), and regions with a gas-liquid mixture corresponding to the re-entrant jet dispersion and vaporisation. The qualitative analysis above is completed with a quantitative comparison shown in Figure 2. The pressure coefficient

\[ c_p = \frac{p - p_{\infty}}{\frac{1}{2} \rho U_{\infty}^2} \]

is plotted against the dimensionless curvilinear abscissa along the body, originating at the axis [16,17].

Second, we compute the cavitating flow over truncated NACA0009 hydrofoil at 2.5° angle of attack and cavitation number \( \sigma = 0.81 \). Extensive experimental investigations were performed by Dupont in his PhD thesis [5], and are used here for validation.

The computational domain considered for the cavitating hydrofoil computations, Figure 3, corresponds to the test section of the cavitation tunnel at the Laboratory of Hydraulic Machines, Ecole Polytechnique Federale de Lausanne.

First, the single phase (liquid) flow is computed. This is done using the two-phase cavitation model, with a large enough cavitation coefficient (\( \sigma = 1.5 \)), such that no cavitation is present. Next, the cavitation coefficient is lowered in small steps (\( \Delta \sigma = 0.1 \)), and the numerical solution is converged after each step. It has been found that by setting directly \( \sigma = 0.81 \)
the solution quickly diverges. This approach is generally recommended when computing cavitating flows, since the flow in the cavity is practically supersonic (due to the low sound velocity within the liquid-vapor mixture), while in the rest of the domain the flow is at zero Mach number (incompressible).

![Graph showing pressure coefficient distribution on the hemispherical fore-body. The curves correspond to the present computations, while the points are experimental data of Rouse and McNown [12].](image)

Figure 2. Pressure coefficient distribution on the hemispherical fore-body. The curves correspond to the present computations, while the points are experimental data of Rouse and McNown [12].

![Diagram illustrating the computational domain for the NACA0009 cavitating hydrofoil.](image)

Figure 3. Computational domain for the NACA0009 cavitating hydrofoil.

![Graph showing pressure coefficient distribution on the NACA0009 hydrofoil. The curves correspond to the present computations, while the points are experimental data of Dupont [5].](image)

Figure 4. Pressure coefficient distribution on the NACA0009 hydrofoil. The curves correspond to the present computations, while the points are experimental data of Dupont [5].
Figure 4 shows a very good agreement between present computations and experimental data.

We have also investigated the effect of the incoming turbulence, using the Singhal model for corrected \( p_r \). As expected, significant differences are observed only at the cavity closure region.

4. THREE-DIMENSIONAL CAVITATING FLOW IN FRANCIS TURBINE RUNNER

The cavitating flow model described and validated in the above sections is further used to investigated a complex three-dimensional flow in the GAMM Francis turbine runner [3,4]. The liquid steady turbulent relative flow in a runner interblade channel is first computed using a mixing interface approach [10].

The operating conditions are set to achieve a cavitation number \( \sigma = 0.1 \). Figure 3 shows the region on the runner blade suction side corresponding to \( \alpha > 0 \). It can be seen that cavitation occurs on the runner blade at the junction with the runner band, where pressure drops below the vaporization pressure.

In order to evaluate the 3D shape and extend of the cavity, we are presenting in Figure 4 the iso-surface of \( \alpha = 0.5 \). Of course, this is only a qualitative assessment of the cavity boundary, as one may choose another iso-surface as the cavity boundary. Nevertheless, the position, shape and size of the cavity seems to be in good agreement with the cavitating flow visualisation in GAMM Francis turbine runner, Figure 5.

Note that the flow visualisation shows a traveling-cloud cavitation, where distinct bubbles can still be observed, Although the mixture model used here does not account for individual bubbles, the fact that \( \alpha \) does not exceed 0.6 inside the cavity shows that there are no parts of the cavity completely filled with vapour.

Figure 3. Vapour volume fraction distribution on the GAMM Francis turbine runner.

Figure 4. Cavity shape at \( \sigma = 0.1 \), presented as an iso-surface of \( \alpha = 0.5 \).

Figure 5. Cavitating flow visualisation for the GAMM Francis turbine, at \( \sigma = 0.14 \), [12].

5. CONCLUSIONS

The paper presents a numerical investigation of cavitating flows using the mixture model implemented in the FLUENT commercial code. The inter-phase mass flow rate is modelled with a simplified Rayleigh equation applied to bubbles uniformly distributed in
computing cells. The main advantage of this approach is that more accurate and reliable cavitation models can be introduced in the FLUENT code via the User Defined Functions. The UDF used in this work is presented in the Appendix.

The cavitation model is validated for the flow around a hemispherical fore-body cavitator, and for the cavitating flow on a NACA0009 hydrofoil. The numerical results agree very well both qualitatively and quantitatively with the experiments.

Next, we investigate the 3D cavitating flow in the GAMM Francis turbine runner at the best efficiency point. The cavity shape and position is in good agreement with the flow visualisation.

**APPENDIX**

The User Defined Function implementation of the interphase mass flow rate in the FLUENT code:

```c
#define MIN_VOF 1.e-5
#define MAX_VOF 0.9999

DEFINE_CAVITATION_RATE(Rayleigh, c, t, p, rhoV, rhoL, vofV, p_v, n_b, m_dot)
{
    real p_vapor = *p_v; /* vapor pressure */
    real n_bubbles = *n_b; /* bubbles per liquid volume */
    real dp, vofM, radV, Rdot;
    dp = p_vapor-ABS_P(p[c],op_pres); /* p_vap - p_abs */
    /* vapor volume fraction restricted to [1.e-5,1.0-1.e-5] */
    vofM=MIN(MAX(MIN_VOF,vofV[c]),MAX_VOF);
    /* bubble radius */
    radV = pow(3.0*vofM/((1.-vofM)*4.0*M_PI*n_bubbles),
                1./3.);
    /* radius time derivative */
    if (dp > 0.0)
        { /* bubble grow */
            Rdot = sqrt(2.*dp/(3.*rhoL[c]));
        }
    else
        { /* bubble collapse */
            Rdot = -sqrt(-2.*dp/(3.*rhoL[c]));
            if (vofV[c]<=MIN_VOF) Rdot=0.0;
        }
    /* compute interphase mass flow rate per unit volume */
    m_dot = rhoV[c]*rhoL[c]*vofM*(1.0-vofM)/
            (vofM*rhoV[c]+(1.0-vofM)*rhoL[c]) *3.0/radV*Rdot;
}
```

**ACKNOWLEDGMENTS**

The authors acknowledge the support from the National University Research Council grants (CNCSIS A 109/2002 and At 220/2003) and Romanian Academy Grant 109/2003. All numerical computations have been performed at the Numerical Simulation and Parallel Computing Laboratory of the “Politehnica” University of Timișoara, National Center for Engineering of Systems with Complex Fluids.

**REFERENCES**


