ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL SIMULATION OF CAVITATION IN TURBOMACHINES USING A PANEL METHOD AND EULER-LAGRANGE FORMULATION

Silveira Luis Victorino
Norberto Mangiavacchi
Universidade do Estado do Rio de Janeiro
silveira_victorino10@hotmail.com
Norberto@uerj.br

Luis M. Portela
Delft University of technology
L.Portela@tudelft.nl

Abstract. *Turbomachines are operational machines that transfer mechanical energy between a rotor and a fluid. One of the main components of a turbomachine responsible for the energy transference, either receiving the rotation of the shaft and transforming it into fluid energy in the case of a pump or transferring energy from the fluid to the shaft in the case of a turbine, is the impeller or rotor. The main objectives of this work were to study the hydrodynamic behavior of the flow in the impeller channels of a turbomachine (radial flow turbopump) using CFD resources and to study the cavitation phenomenon occurring in the suction region of a turbomachine working with a liquid. The objective of the hydrodynamic analysis was to study the flow behavior around the impeller blades by analyzing the velocity profile and pressure distribution by numerical simulation. A vortex based panel method, which is a very efficient method for the analysis of the potential flow around hydrodynamic profiles, was adapted for the simulation of flows in turbomachines. An Euler-Lagrange approach, coupled with the Langevin equation model and the Rayleigh-Plesset model, was employed to estimate the bubble trajectory and radius, in the impeller of a turbomachine. Results of the simulations show the regions of bubble collapse for different flow and geometric parameter choices, and provide insights for improvements on cavitation prevention in turbomachines.*

Keywords: *Numerical simulation, Computational fluid dynamics (CFD), Turbomachines, Cavitation*

1. INTRODUCTION

In liquid flows, cavitation occurs if the local pressure drops below the saturated vapor pressure, which leads to the formation of vaporous bubbles to relieve the negative pressure (Batchelor (1967)). Cavitation is commonly observed when a hydraulic machines operates in high speeds or under design conditions. It can lead to problems such as pressure pulsations, sudden changes in loads, vibrations, noise and erosion (Brennen (1995).

Phenomenologically, cavitation often involves complex interactions between turbulent flow structures and phase change dynamics (Tseng and Shyy (2010)). The cavitation phenomenon involves two-phase flow: the liquid to be pumped and the vapor bubbles which are formed during pumping and which travel in the turbomachine until their collapse. The formation process of these bubbles is complex, but occurs mainly due to the presence of regions of very low pressure. The collapse of the bubbles can often lead to a deterioration of the material, depending on the intensity or speed of bubble collapse.

The numerical simulation of multiphase flows is one of the most challenging CFD (computational fluid dynamics) problems occurring in many engineering applications. Among them are cavitating flows, in which a liquid flow vaporises in regions where the pressure drops below the vapor pressure.

The main aim of this work is to show the cavitation bubbles dynamics using the Eulerian-Lagrangian approach. In this approach a continuum description is used for the liquid phase (Eulerian manner), the bubbles are modelled as individual spherical particles, bubble transport is solved from a bubble equation of motion and discrete tracking of the bubbles (Lagrangian manner). In Eulerian manner, the flow variables are a function of space and time and thus are represented as fields. In the Lagrangian manner instead individual particles are considered and the position and velocity of each particle is a function of time only. The bubbles are usually modelled as spherical point-particles with models for fluid-bubble interaction forces and bubble-bubble interactions, the vapor-volume fraction is obtained by simulating the evolution of individual bubbles, composing a discrete phase (Darmana and Kuipers (2006), Ferrante and Elghobashi (2007)). The dynamics of these bubbles is computed using Newtonian equations of motion coupled with Rayleigh-Plesset equation (RP) for description of bubble size. This approach considers each bubble individually, it allows to take into account various forces acting on the bubble, as well as bubble-wall interactions, inhomogeneous and transient water quality effects such

ISSN 2176-5480

as bubble spectra and non-condensable gas content. The flow of a liquid or of a liquid-vapor mixture carries the tracked vapor bubbles (Wörner (2003)). The state of the art of the method can be found in (Crowe and Sommerfeld (1998)).

A vortex based panel method, which is a very efficient method for the analysis of the potential flow around hydrodynamic profiles, was adapted for the simulation of flows in turbomachines. The panel method are useful tool and economical computationally for analysing and projecting of aerodynamics profiles. In this work the classical Hess and Smith (1967) method was used, which is based on uniform distribution of sources (variables) and vortex (constant) and in Neumann's boundary condition, being one of most known and used by aerodynamicists. However, it is a method that usually produce spurious loads in the region of a tapered trailing edge, as in that the attack angle and camber of the airfoil increase. The main cause is the constancy of the vortices intensity, proportional to the circulation and whose increasing difficult consistent application of the Kutta condition. Those problems can be mitigated by the use of variable distributions of vortices with zero value at the trailing edge (Plotkin (1990);Girardi and Bizarro (1995); Petrucci (1998)) and also by the use of distribution sources order higher and also by using order higher source distributions (Hess and Smith (1967)).

2. MODEL DESCRIPTION

2.1 Mathematical formulation

In the Euler-Lagrange modelling, the continuous phase is solved by Euler framework by solving the Navier-Stokes equations in the computational domain. The dispersed phase is simulated considering particles affected by forces from the continuous fluid, applying these forces on each particle using the Newton's second law we get the acceleration of the dispersed phase particle through the continuous phase. When only the effect of the continuous phase is considered on the dispersed phase, it is called one-way coupling regime (Dukowicz (1980)). When both the effect of dispersed phase on the continuous one and vice versa is considered then it is called Two-way coupling regime; in two-way coupling regime, the effect of the bubbles on the continuous phase is considered by introducing source terms in the Navier-Stokes equations and changing the volume available for the continuous phase in the computational cells according to the void fraction in the cell (Sommerfeld (2000)). The Eulerian-Lagrangian numerical simulation methods have been developed with different assumptions. In problems such as the dispersion of atmospheric pollutants, it may be assumed that the particles do not perturb the flow field. The solution then involves tracking the particle trajectories in a known velocity field i.e. the fluid phase equations are solved independent of the particles (Gauvin and Knelman (1975)). In other problems the particles may carry sufficient momentum to set the surrounding fluid in motion. In this case it is necessary to include the fluid-particle momentum exchange term in the fluid phase equation. However, the volume occupied by the particles in a computational cell in comparison with the volume of the fluid may still be neglected (Crowe and Sharma (1977)). When the particle volume is significant it is important to model the volume fraction in both the momentum and continuity equations.

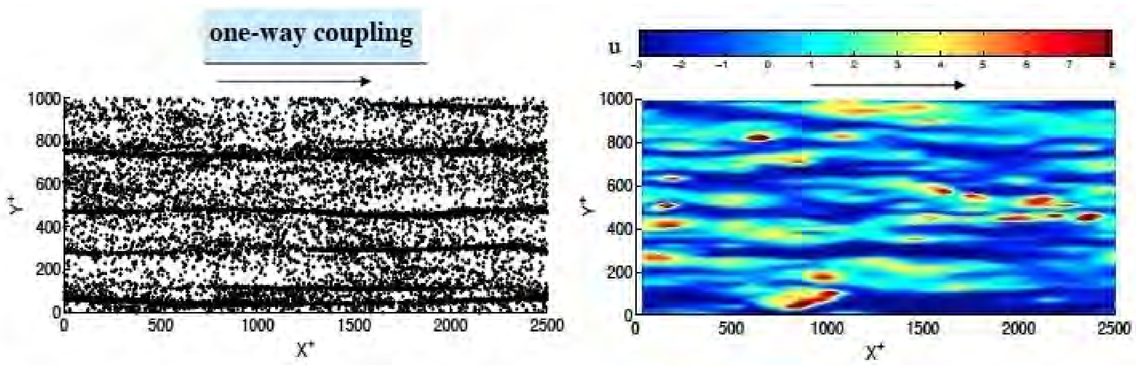


Figure 1. Particles moving up and down the wall (Rashidi (1990))

Numerical schemes based on mathematical models of separated particulate multiphase flow have used the continuum approach for all the phases or a continuum approach for the fluid phase and a Lagrangian approach for the particles. These simulation methods can be applied in various settings; e.g. sedimenting and fluidized suspensions, lubricated transport, hydraulic fracturing of reservoirs, slurries, sprays, etc.

Exist another formulation, the Eulerian-Eulerian approach, that considers the particulate phase to be a continuous fluid interpenetrating and interacting with the fluid phase (Gidaspow (1994)), which is not our aim in this work.

2.1.1 Lagrangian solver

In these work one-way coupling is adopted, which takes only the effect of the continuous phase on the dispersed one. One-way coupling has the significant advantage that the Eulerian velocity field can be computed independent of the particle tracking by a standard single-phase simulation. For a steady flow for example, the continuous phase velocity field

ISSN 2176-5480

can be obtained once at the beginning. The trajectories of the individual particles can then be computed independently from one another. The effect of the continuous phase is taken by calculating the relative forces from the continuous phase acting on the dispersed one. These forces are drag force, Lift force, virtual mass force, buoy force, pressure force and others. Applying these forces in Newton's second law for each bubble, the acceleration of the bubble each time step of the Lagrange simulations can be expressed as follows:

$$\left(\frac{d\mathbf{x_b}}{dt}\right) = \mathbf{U_b} \tag{1}$$

$$\left(\frac{d\mathbf{U_b}}{dt}\right) = \mathbf{a} \tag{2}$$

$$M_b\left(\frac{\partial\mathbf{U_b}}{\partial t}\right) = \sum\mathbf{F_b} = \mathbf{F_{drag}} + \mathbf{F_{lift}} + \mathbf{F_{press}} + \mathbf{F_{buoy}} + \mathbf{F_{AddedMass}} \tag{3}$$

Where $\mathbf{U_b}$ is the velocity and M_b the mass of the particle. $\sum\mathbf{F_b}$ is the sum of force exerted by the fluid on the particle, $\mathbf{F_{drag}}$ is drag force, $\mathbf{F_{lift}}$ is lift force, $\mathbf{F_{press}}$ is pressure gradient force due field pressure gradients, $\mathbf{F_{buoy}}$ is buoyancy force and $\mathbf{F_{AddedMass}}$ is added-mass force.

$$\mathbf{F_{drag}} = \frac{1}{2}\rho_l A_b C_D|\mathbf{U_b} - \mathbf{U}_l|(\mathbf{U_b} - \mathbf{U}_l) \tag{4}$$

$$\mathbf{F_{lift}} = m_l C_L(\mathbf{U_b} - \mathbf{U}_l) \times \nabla \times \mathbf{U}_l \tag{5}$$

$$\mathbf{F_{buoy}} = (m_b - m_l)\mathbf{g} \tag{6}$$

$$\mathbf{F_{Addedmass}} = -\frac{1}{2}m_l\left(\frac{D\mathbf{U_b}}{Dt} - \frac{d\mathbf{U}_l}{dt}\right) \tag{7}$$

$$\mathbf{F_{Press}} = -V_b\nabla\mathbf{p} \tag{8}$$

ρ_l is the liquid phase density (carrier phase), A_b is the bubble area, C_D is the coeficient of bubble drag, $C_L = \frac{3.1}{\sqrt{Re_b}}$ is the is coeficient of bubble lift, $\mathbf{U_b}$ is the bubble velocity, $\mathbf{U}_l$ is the liquid phase velocity, m_l is the liquid phase mass, m_b is the bubble mass, $\frac{D\mathbf{U_b}}{Dt}$ is a material derivative of bubble velocity, ∇p is the gradient pressure and V_b is the bubble volume.

Jr Johnson and Hsieh (1966) performed Lagrangian simulations of cavitating bubbles traveling around a blunt body and included the drag force with a drag coeficient determined by Habermann and Morton (1953) and also the contribution of volume change of the bubble in time to the added mass force. Thomas and Hunt (1984) included contributions due to lift, with a constant lift coeficient of 1/2. Auton and Prud'homme (1988) showed that the constant lift coeficient of 1/2 is appropriate in inviscid limit, and showed that for forces due to added mass and fluid accelaration, the material derivative is appropriate term for the fluid accelaration.

The expression for the coeficient of bubble drag, a function of bubble Reynolds number and determinated experimentally by Habermann and Morton (1953), is:

$C_D = \frac{24}{Re_b}(1 + 0.15{Re_b}^{0.687})$ for $Re_b < 1000$ and $C_D = 0.44$ for $Re \geq 1000$

Where the bubble Reynolds number is defined as $Re_b = \frac{2R_b(\mathbf{U_b}-\mathbf{U}_l)}{\nu}$ is the bubble Reynolds number. For small bubbles in the water, the drag profile is similar to the solid spheres $C_D \sim \frac{24}{Re_b}$ due to the contamination of the bubble surface.

The integration of the equation Eq. (3), requires very small discrete time steps and is rather expensive for large number of bubbles.

For the particle tracking and trajectory computation of the particles in the flow it was assumed that the unsteady velocity field of the continuous phase $\mathbf{U}_l$ is given by the potential flow on the channel or blade passage, modified by a one-dimensional RANS solution, plus a random turbulent fluctuation. Considering that the particles can be subject to different forces (Drag, Lift, Added mass, pressure, etc), some of them are neglected based on order of magnitude estimates. We are also neglecting the influence of the dispersed phase on the continuous phase and inter-particle interactions (i.e. assume one-way coupling) too. The code should be able to deal with arbitrary initial conditions for the particles (for both the position and the velocity), prescribed as an input.

2.1.2 Euler solver

General speaking, Euler framework uses the Reynolds Averaged Navier-Stokes equations (RANS) for conservation of mass, momentum and Energy basis through the computational cells for solving any problem of fluid dynamics and heat transfer. In the present work, we are concerned only with the two phase flow matter at ambient temperature with phase change given by the Rayleigh-Plesset model.

In the mathematical treatment we solve a set of continuity and momentum equations for the primary phase only and for the secondary/dispersed phase the trajectories (of dispersed phase) are calculated by using the equation of motion. For this we use Newton's law of motion as above said, that is just a force balance taking into account the interaction between the primary and secondary phase. Below are shown the equations applied while modeling flows with the Euler-Lagrange approach. We are dealing with a very small volume-fraction of bubbles and the effect of the bubbles on the continuity equation can be neglected.

The conservation equations in the present case differ a little from the normal fluid dynamics equations as there are two phases in the same cell. So, the effect of the dispersed phase when writing the conservation equations should be considered. The interaction between the particles and the fluid is felt through an exchange of momentum. The continuity and Navier-Stokes equations for the continuous phase become:

$$\nabla \cdot \mathbf{U} = 0 \tag{9}$$

$$\rho \left(\frac{\partial \mathbf{U}}{\partial t} + (\nabla \mathbf{U}) \cdot \mathbf{U} \right) = -\nabla P + \mu \nabla^2 \mathbf{U} + \mathcal{F} \tag{10}$$

Where $\mathbf{U}$ is the velocity of the fluid (continuous phase), P is the pressure, ρ is the density, and μ is the viscosity. The continuity equation is exactly the same as for an incompressible flow without particles. The Navier-Stokes equation has the extra term $\mathcal{F}$, which is the force per unit of volume due to the particles. If the bubble mass concentration is very small, then $\mathcal{F}$ can be neglected in Equation Eq. (10). The continuity and Navier-Stokes equations become exactly the same as for the flow without bubbles presence, and Eq. (9) and Eq. (10) are uncoupled. In addition to the solution of the single phase flow, the problem simply requires the use of an algorithm for tracking the individual bubbles (one-way coupling). If $\mathcal{F}$ cannot be neglected in Eq. (10), then Eq. (9) and Eq. (10) need to be solved simultaneously (two-way coupling). Equation (9) and Eq. (10) are solved together with the equations for the trajectory of the particles.

For liquid phase modelling we have been using the eddy-viscosity Prandtl mixing-length model for a developed flow in a unsteady fully-developed single-phase channel flow using the finite-volume method. Two possible wall boundary conditions are considered: (i) prescribed velocity at the wall and (ii) prescribed velocity-gradient at the wall. Currently, we solve the two-dimensional potential flow in the channel or blade passage by a vortex based panel method, and solve Eq. (10) for a one-dimensional flow to get the turbulent boundary layer profile.

2.2 Rayleigh-Plesset model

The dynamics of spherical vapor bubbles, expressed by the Rayleigh-Plesset equation, has been extensively studied following the original works of Rayleigh (1917) and Plesset (1964). The teory was summarized in a differential equation for the bubble radius R(t) by Gilmore (1952) and extended and refined by many other reseachers. Early experimental and analitical studies are done by Habermann and Morton (1953) and others.

The Rayleigh-Plesset equation models, the temporal growth and collapse of spherical vapor bubbles in isothermal environment. Gas diffusion and heat transfer effects are neglected. The isentropic law is assumed for non-condensable gas inside the bubble. Essential modifications were added by Hsiao and Chahine (2004) to account for a slip velocity between the bubble and the host liquid and for the consideration of a non-uniform pressure field along the bubble surface, called surface-averaged pressure equation. As formulated in Chahine (2008), have been considered a spherical bubble of radius $R(t)$, which can change with time (t) in an infinite domain of a liquid. When the temperature in the domain is constant and the liquid is incompressible, so, generalized equation describes the motion of the bubble wall; resulting differential equation for the bubble dynamics, reads:

$$\rho_l \left[R_b \frac{d^2 R_b}{dt^2} + \frac{3}{2} \left(\frac{dR_b}{dt} \right)^2 \right] = p_b - p_\infty - \frac{2\sigma}{R_b} - \frac{4\mu_l}{R_b} \frac{dR_b}{dt} \tag{11}$$

Where R_b is the bubble radius, p_b and p_∞ are the pressure inside and outside of the bubble, σ is the surface tension coeffient, and μ_l and ρ_l are the liquid viscosity and density, respectively. To estimate p_b, it is typically assumed that the bubble contains some contaminant gas which expands or contracts according to adiabatic or isothermal processes (Brennen (1995) and Chahine and Hauwaert (1993)). The bubble inside pressure (p_b) consists of contribution from the gas pressure p_g and the vapor pressure p_v.

ISSN 2176-5480

3. RESULTS

3.1 Turbulent flow in on a single-phase channel

The first validation results are obtained by solving the RANS equations for the developed flow on a channel. Figure 2 shows the velocity profile obtained employing the eddy-viscosity Prandtl mixing-length model. The obtained velocity profile is consistent with theoretical results (Antal and Flaherty (1991) and Bonakdari and Joannis (2008)).

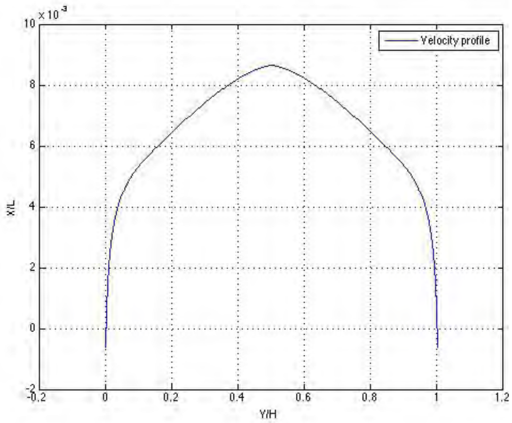


Figure 2. Velocity profile in a unsteady fully-developed single-phase channel flow with variable viscosity

3.2 Turbulent bubbly flow on a channel

In this section results from bubble simulations on a channel are presented, for two different bubble densities, obtained by solving Eq. (3). The simulations fig.(3) e fig.(4) show that to obtain statistically converged mean values a large number of bubbles is required. This a shortcoming of the method.

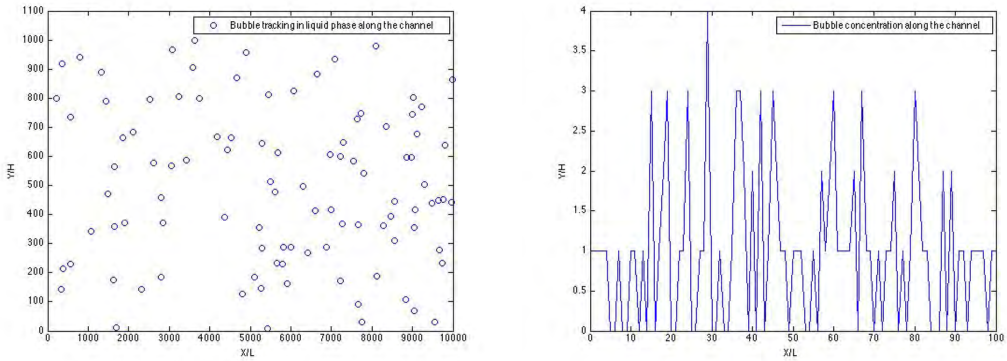


Figure 3. Bubble simulation of both phases and spacial distribuition of bubbles concentration in the channel (simulation with 100 bubbles)

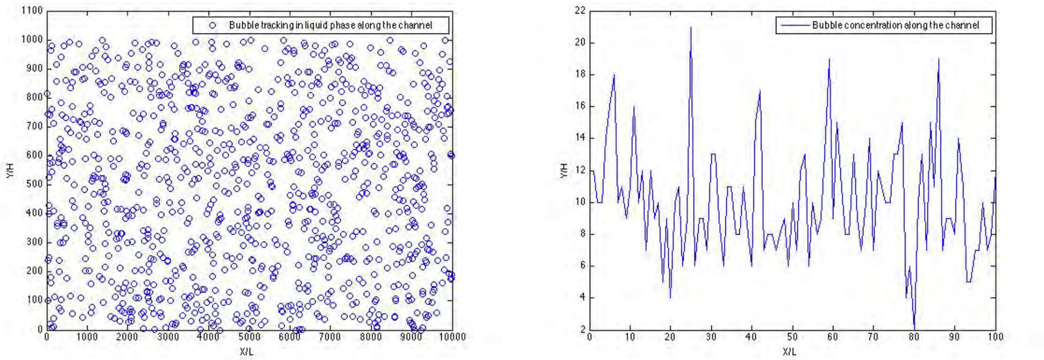


Figure 4. Bubble simulation of both phases and spacial distribuition of bubbles concentration in the channel (simulation with 1000 bubbles)

3.3 Flow on a turbomachine passage

The method is applied to study the behavior of bubbles in a passage between blades on a turbomachine. The flow velocity field is obtained by combining the two-dimensional potential flow obtained using a vortex based panel method, and the turbulent velocity profile from the single phase one-dimensional RANS simulation, with pseudo-random velocity fluctuations, as shown in fig.(5).

An example for the bubble positions on a blade passage is shown in Fig. 5 for two bubble densities. As the number of simulated bubbles is increased, the accumulation of bubbles on the surfaces of the blades becomes more pronounced, as expected by the effect of turbophoresis.

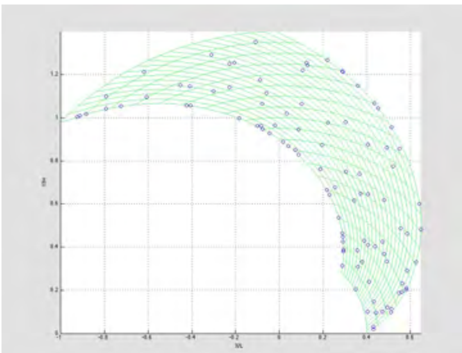
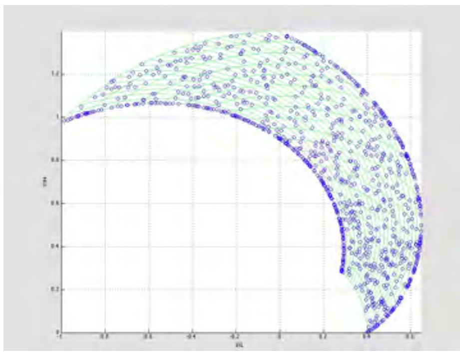
a)  b) 

Figure 5. Typical bubble distributions on a turbomachine passage between two blades. Left: simulation with 100 bubbles. Right: simulation with 1000 bubbles

4. Conclusions

The validation results herein presented show that the proposed scheme reproduces results that are qualitatively consistent. The developed method is suitable for tests of bubble dynamic models developed employing more costly direct numerical simulations of bubble flows. Finally, this approach (Euler-Lagrange) can show all the variables of the moving bubbles instantaneously. This includes the ability to have information about the different forces acting on each bubble, change of velocity and velocity fluctuations of the bubbles, etc. This approach can be considered a smart tool for testing different models for the physical behavior of the bubbles moving in a continuous phase.

Further development of the numerical code will be focused to the implementation of various models for the turbulent velocity, and drag and lift forces acting on the bubbles.

5. ACKNOWLEDGEMENTS

Support from CAPES scientific agency is acknowledged.

ISSN 2176-5480

6. REFERENCES

Antal, S. and Flaherty, J., 1991. "Bubbly flow in a vertical channel". *Int. Journal of Multiphase Flow*, Vol. 17, pp. 635–652.

Auton, T.R. Hunt, J. and Prud'homme, M., 1988. "The force exerted on body in inviscid unsteady non-uniform rotational flow". *J. Fluid Mech.*, Vol. 257, pp. 197–241.

Batchelor, G., 1967. *An introduction to fluid mechanics*. Cambridge university Press.

Bonakdari, H. and Joannis, C., 2008. "Turbulent velocity profile in fully-developed open channel flows". *Environ. Fluid Mech.*, pp. 4–7.

Brennen, C.E., 1995. *Cavitation and Bubble Dynamics*. Oxford university Press, New York, 1st edition.

Chahine, G., 2008. "Numerical simulation of bubble flow interaction". *In: Proceedings of 2nd International cavitation Forum, WIMRC, University of Warwick, great britain*, Vol. 1, pp. 72–86.

Chahine, G. and Hauwaert, P., 1993. *Study of interaction between a bubble and a vortical structure*. ASME-PUBLICATIONS-FED, Boston.

Crowe, C. and Sommerfeld, M., 1998. "Multiphase flows with droplets and particles". *CRC Press, Boca Raton*, pp. 35–37.

Crowe, C. and Sharma, M., 1977. "The particle-source-in cell model for gas-droplet flows". *Trans. ASME J. Fluids Engrg.*, Vol. 99, pp. 325–332.

Darmana, N. and Kuipers, J., 2006. "Parallelization of an euler-lagrange model using mixed domain decomposition and a mirror domain technique: Application to dispersed gas-liquid two-phase flow". *J. Comput. Phys.*, Vol. 1, pp. 220–248.

Dukowicz, J., 1980. "A particle-fluid numerical model for liquid sprays". *Phys.*, Vol. 35, pp. 209–253.

Ferrante, A. and Elghobashi, S., 2007. "On the accuracy of the two-fluid formulation in direct numerical simulation of bubble-laden turbulent boundary layers". *J. Phys. Fluids*, Vol. 4, pp. 19–19.

Gauvin, W. and Knelman, F., 1975. "Drop trajectory predictions and their importance in the design of spray dryers". *Int. J. Multiphase Flow*, Vol. 1, pp. 793–816.

Gidaspow, D., 1994. *Multiphase Flow and Fluidization Continuum and Kinetic Theory Descriptions*. Academic press, Boston.

Gilmore, F., 1952. "The growth and collapse of a spherical bubble in a viscous compressible liquid". *California Institute of Technology*, Vol. 1, pp. 24–26.

Girardi, R.M. and Bizarro, A.F., 1995. "Modification of the hess and smith method for calculating cascades and airfoils with cusped trailing edge". *13th Brazilian Congress and 2nd Iberian American Congress of Mechanical Engineering*.

Habermann, W. and Morton, R., 1953. *An experimental investigation of the drag and shape of air bubbles rising in various liquids*. Academic press, Boston.

Hess, J. and Smith, A., 1967. "Calculation of potential flow about arbitrary bodies". *Progress in Aeronautical Sciences*, Vol. 8, pp. 1–138.

Hsiao, C.T. and Chahine, G., 2004. "Prediction of vortex cavitation inception using coupled spherical and non-condensable models and navier-stokes computations". *Journal of Marine Science and Technology*, Vol. 8, pp. 99–108.

Jr Johnson, V. and Hsieh, T., 1966. "The influence of the trajectories of gas nuclei on cavitation inception". *6th Naval Hydrodynamics Symp.*, Vol. 1, pp. 163–179.

Petrucci, D., 1998. *Problema Inverso do Escoamento em Torno de Perfis AerodinÃćmicos Isolados e em Grades de TurbomÃąquinas*. Ph.D. thesis, DissertaÃğÃčo de Mestrado, EFEI, Brazil.

Plesset, M., 1964. "Bubbles dynamics in cavitation in real liquids". *Elsevier Publishing Company*, Vol. 1, pp. 1–17.

Plotkin, A., 1990. "Comment on improved thin-airfoil theory". *Journal of Aircraft*, Vol. 27, pp. 479–479.

Rashidi, G. H. M.; Benerjee, S., 1990. "Particle-turbulence interaction in a boundary layer". *Int. J. Multiphase Flow*, Vol. 16, pp. 935–949.

Rayleigh, L., 1917. "On the pressure developed in a liquid during collapse of a spherical cavity". *Philosophical Magazine*, Vol. 34, pp. 94–98.

Sommerfeld, M., 2000. "Teorical and experimental modelling of particulate flows - overview and fundamentals". *VKI Lecture series*, Vol. 6, pp. 1–62.

Thomas, N.H. Auton, T.S.K. and Hunt, J., 1984. "Entrapment and transport of bubbles by plunging water". *D. Reidel Publishing Co.*, Vol. 1, pp. 255–268.

Tseng, C. and Shyy, W., 2010. "Modelling for isotermal and criogenic cavitation". *Int. Journal of heat and mass transfer*, Vol. 53, pp. 513–525.

Wörner, M., 2003. *A compact introduction to the numerical modelling of multiphase flows*. Ph.D. thesis, Karlsruhe institute of technology, Germany.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.