HAL
open science

Numerical modeling of a two-dimensional aerated cavitation in a symmetrical venturi nozzle

P Tomov, S Khelladi, Florent Ravelet, C Sarraf, F Bakir, D Giroux

To cite this version:

P Tomov, S Khelladi, Florent Ravelet, C Sarraf, F Bakir, et al.. Numerical modeling of a two-dimensional aerated cavitation in a symmetrical venturi nozzle. 22è Congrès Français de Mécanique, Aug 2015, Lyon, France. hal-01189371

HAL Id: hal-01189371

https://hal.science/hal-01189371

Submitted on 1 Sep 2015

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.

Numerical modeling of a two-dimensional aerated cavitation in a symmetrical venturi nozzle

P. TOMOV[a,b], S. KHELLADI[a], F. RAVELET[a], C. SARRAF[a], F. BAKIR[a], D. GIROUX[b]

a. DynFluid Laboratory, Arts et Métiers ParisTech
151, Boulevard de l'Hôpital, 75013 Paris, FRANCE
b. SNECMA, SAFRAN group,
Rond-point Réné Ravaud, 77550 Réau, France
Petar.TOMOV@ensam.eu

Keywords: Aerated cavitation, Homogeneous Equilibrium Model, LES, Finite-Volume Moving Least Squares, Penalization method, Venturi nozzle

Abstract :

Cavitation is a well-known physical phenomenon occurring in various technical applications. Its coupling with the aeration, is a recent technique, which allows the control of the overall effect of the cavitation. The aeration is achieved by introducing air bubbles into the flow. In order to reveal and explore the behaviour of air in the vicinity of the cavitation regions, the paper is oriented towards the physics of the colliding vapour phase in the presence of cavitation. By penalizing the strain rate tensor in the Homogeneous Equilibrium Model, a two-way cavitation-aeration coupling is achieved. The contact-handling algorithm is based on the projections of the velocity fields of the injected bubbles over the velocity field of the fluid flow. At each time step the gradient of the distance between the bubbles, is kept non-negative, as a guarantee of the physical non overlapping. The bubbles are considered as non-deformable. The differential equations system is composed of the 2D Navier-Stokes equations, implemented with the Homogeneous Equilibrium Model. A high-order Finite Volume solver based on Moving Least Squares approximations is used. The code uses a SLAU-type Riemann solver for the accurate calculation of the low Mach numbers. The computational domain is a symmetrical 2D venturi nozzle, with 18° - 8° convergent/divergent angles respectively.

Introduction:

Cavitation is an extremely wide spread physical phenomenon in various technical applications. When the pressure becomes inferior to the saturating vapour pressure of

the liquid, cavitation takes place. It is responsible for issues like erosion [1], noise and vibrations [2, 3], which can lead to a malfunctioning of various turbo-machines. In general, the presence of cavitation has a negative effect on the normal functioning of a hydraulic system. Nevertheless, in some particular cases, it can have an extremely positive effect leading, for instance into a drag reduction, as it is the case of submarine vehicles [4], where the supercavitation covers the immersed body and makes it slip through the liquid [5]. It is very important for one to understand the physics behind the complex two-phase flow phenomenon, in order to reduce the negative effect or increase its positive influence. Studying cavitation dynamics in simple geometries like convergent-divergent venturi nozzles is a way of achieving that goal. On the top of that, those type of obstacles lead to a rich sheet cavitation dynamics [6]. In the case of the venturi nozzle, a periodic cycle with the appearance of a re-entrant jet can take place. In general, the re-entrant jet is created by the flow which expands in the closure region, in such a way, that in combination with the venturi wall, it creates a stagnation point. The conservation of momentum makes the fluid to pass beneath the cavity. As a result, the jet progresses and results in a vapour separation [7], which forms a cloud which is being further advected down the stream. The cloud vapour collapses in the divergent venturi nozzle zone where the pressure is substantially higher than the one at the throat. By its nature, the cloud cavitation has an extremely aggressive behaviour and it is capable of doing severe damage on the near structure. Hence, a control of the cavitation behaviour can lead to a stable regime instead of having an unsteady damaging one.

A recent technique capable of influencing the cavitation inception is the aeration of the liquid. Davis et al. [8] and later Dunn et al. [9] injected a controlled quantity of bubbles into a transparent venturi nozzle, in order to study its effects on the cavitation in the case of water and aviation fuel. It has been found that the cavitation inception place can be shifted forward or backward, if an injection of gas is to take place or not. Coutier et al. [10] measured the speed of sound in a two-phase flow which was characterized by a high void fraction. In order to achieve such a high quantity of gas, an intrusive injection of air into a liquid flow has been done. Dong et al. [11] presented an investigation of cavitation control by aeration. The pressure waveforms were analyzed with and without aeration. The results showed that aeration phenomenon increases in a remarkable manner the pressure in the cavitation region and the corresponding pressure waves exhibit a shock wave.

Numerical Method:

From a numerical point of view, the main difficulty in cavitating flows lies in the treatment of two very different from each other regions: the first one, which is pure liquid, is nearly incompressible flow, and the second one is the low-viscosity vapour region. Moreover, a special care must be taken for the transition region, which is not always clearly distinguished. Appropriate high-resolution mesh densities and time steps of the order of 10^{-7}s [12], are necessary in order to obtain proper convergence and rep-

resentation of the physical phenomenon. Unfortunately, the computer power, required to capture all the processes occurring over a wide range of time and length scales is beyond the present capabilities, therefore, simplifications and precise approaches are necessary to obtain a realistic model to simulate cavitating flows. Moreover, the injection of multiple single bubbles into the multiphase cavitating flow makes the simulation procedure even more delicate and complex. Their presence may impact the cavitating zones' inception, shape and shedding frequency. As a result the aeration-cavitation coupling is of crucial importance for the proper representation of the flow dynamics.

The present study deals with the aeration-cavitation coupling in a horizontal symmetric venturi nozzle. The numerical modelling of the cavitation phenomenon is achieved by the Homogeneous Equilibrium Model (HEM) [19]. The differential equation system is composed of the 2D Navier - Stokes (NS) equations coupled with the HEM. It accounts for the momentum, mass and energy equations. In the cavitation zone, the conservation equations tend to lose locally their hyperbolic character and become elliptic, therefore the liquid and vapour phases are separately described by equations of state. As a result, the two phases keep proper thermodynamic behaviour [20]. The thermodynamic equilibrium is present at each point of the computational domain, therefore, the vapour phase pressure is considered as constant and it is equal to the saturation vapour pressure. The model does not take into account the relative motion between the phases.

The conserved variables are calculated in the cell centres. By using Taylor expansion series, any variable can be expressed in terms of its successive derivatives through a weighted least-squares fitting. A major difficulty is the calculation of successive derivatives. This issue is overcome by using the Moving Least Squares (MLS) approach in a Finite-Volume (FV) framework [21]. Such a configuration does not introduce new degrees of freedom. Another advantage is the good performance on unstructured grids [22]. In order to implement the MLS approach, a number of neighboring points (stencil) of each node ought to be defined. Its construction is of crucial importance for the behaviour of the numerical simulation [23]. Moreover, the numerical approach needs to deal with shocks and strong gradients. As a result, slope limiters [24] are coupled with detector gradient [25]. The simulation belong to the family of the implicit Large Eddy Simulation (iLES). The used turbulence model is based on the one developed in [23].

The Riemann (problem) solver is of capital importance, due to the large spectrum of different coexisting flow regimes. The flow is subsonic in the liquid (very low Mach) and vapour (intermediate Mach) pure phases, and supersonic in the mixture phase [26]. In order to cope with this issue, the FV-MLS code uses a modified Simple Low dissipative Advection Upstream splitting method (SLAU) [27].

Bubble Injection Approach:

The aerated phenomenon is simulated by a controlled injection of bubbles into the multiphase flow. Each bubble is treated individually and the NS equations are solved for the moving fluid. The contact - handling algorithm is based on the projection of the velocity field of the injected bubbles over the velocity field of the fluid flow. The injection is done in such a manner that, at each time step the gradient of the distance between every two bubbles, is kept non-negative as a guarantee of the possible non-overlapping [28, 29]. The method consists of imposing a constraint on the velocity field of the bubbles, as a guarantee that at each time step the calculated bubble velocity field belongs to an eligible velocity field of the fluid. The numerical approach takes into account the gravity effects. The motion of each bubble is governed by Newton's second law. The forces acting on each bubble are the buoyancy force, the lift and drag forces and a turbulence force [30, 31]. The code takes into account the force as a result of surface variation, since the bubbles are treated as non-deformable.

A penalization procedure is applied, in order to achieve the two-way coupling between the multiphase flow and the bubbles. The fluid is considered as a Newtonian, hence the stress tensor is proportional to the strain rate. A weak formulation of the problem over the hole computational domain is needed, since a re-meshing technique has not been applied. A characteristic function is defined, which takes the form a equation of a circle. The calculations take place either inside or outside the bubble. As a result, a penalization method is obtained and it is applied to the viscosity term. When the limit of the stress tensor equals zero, the bubbles stay rigid through their motion. Otherwise, when a bubble is near or inside a cavitating zone, the viscosity will be penalized and take the value of the mixture viscosity [32].

The domain of calculation is an axisymmetrical venturi duct (fig.1) with an 18° - 8° convergent/divergent angles, respectively. The total length is 220 mm, and the inlet/throat section ratio is equal to 3. The liquid used in the simulation is water at

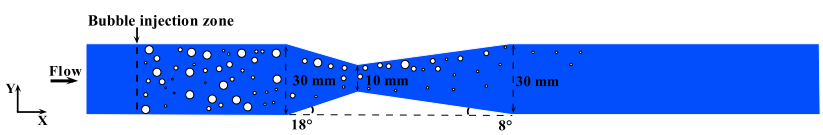

Figure 1: venturi

$T = 300$ K. The saturation vapor pressure in operating conditions is taken $P_{vap} \approx 2200$ Pa. Due to the pressure shock waves present in the multiphase flow [33], the numerical domain is extended (once upstream and once downstream) in longitudinal direction. Absorbing boundary conditions are imposed [12], in order to evacuate the upfront coming pressure waves. The flow velocity at the inlet section is set to be 4 m/s and the pressure outlet is equal to 50 kPa. The initial distribution of the bubbles is achieved by a random function. The size and the quantity of the bubbles are initially defined. The work is the logical continuation of the numerical study presented in [34].

References

[1] M. Petkovšek, M. Dular, Simultaneous observation of cavitation structures and cavitation erosion, Wear 2013, vol.300, pp.55–64.

[2] Y. Tsujimoto, K. Kamijo, C.E. Brennen, Unified treatment of flow instabilities of turbomachines, Journal of Propulsion and Power 2001, vol.17(3), pp.636-643.

[3] Y. Tsujimoto, Flow Instabilities in Cavitating and Non-Cavitating Pumps, Educational Notes RTO-EN-AVT-143, 2006.

[4] M. Wosnik, R.E.A Arndt, Measurements in High Void-Fraction Bubbly Wakes Created by Ventilated Supercavitation, Journal of Fluids Engineering 2013, vol.135.

[5] S.L. Ceccio, Friction Drag Reduction of External Flows with Bubble and Gas Injection, Annual Review of Fluid Mechanics 2010, vol.42, pp.183–203.

[6] R. Campos, Analyse des ecoulements cavitants stationnaires et instationnaires dans les turbomachines, PhD Thesis, 2009.

[7] R.B. Wade, A.J. Acosta, Experimental Observations on the Flow Past a Plano-Convex Hydrofoil, Journal of Basic Engineering 1966,vol.88, pp.273-282.

[8] M.P. Davis, Experimental Investigation of the Aviation Fuel Cavitation, Phd Thesis, 2008.

[9] P.F. Dunn, F.O. Thomas, M.P. Davis, I.E. Dorofeeva, Experimental characterization of aviation-fuel cavitation, Physics of Fluids 2010, vol.22(11), 117102.

[10] O. Coutier-Delgosha, J.F. Devillers, T. Pichon, A. Vabre, Internal structure and dynamics of sheet cavitation, Physics of Fluids 2006, vol.18.

[11] Z.Y. Dong, P.I. Su, Cavitation Control By Aeration and Its Compressible Characteristics, Journal of Hydrodynamics, Ser B, 2006, vol.18(4), pp.499–504.

[12] H. Gunter, G.H. Schnerr, H.I Sezal, S.J. Schmidt, Numerical investigation of three-dimensional cloud cavitation with special emphasis on collapse induced shock dynamics, Physics of Fluids 2008, vol.20(4), 040703.

[13] Y. Chen, S.D.Heister, A numerical treatment for attached cavitation, Journal of fluid and engineering 1994, vol.116, pp.613–618.

[14] M. Deshpande, Numerical modeling of the thermodynamic effects of cavitation, Journal of fluids and engineering 1997, vol.119, pp.420–427.

[15] W. Dijkhuizen, Deriving closures for bubbly flows using direct numerical simulations, PhD Thesis, 2008.

[16] G.H. Schnerr, J. Sauer, Physical and Numerical Modeling of Unsteady Cavitation Dynamics, Proceedings of 4th International Conference on Multiphase Flow, New Orleans 2001, USA.

[17] Y.C.Wang and C.E.Brennen, Numerical computation of shock waves in a spherical cloud of cavitation bubbles, Journal of Fluids Engineering 1999, vol.121, pp.872–880.

[18] C.F. Delale, G.H. Schnerr, J. Sauer, Quasi-one-dimensional steady-state cavitating nozzle flows, Journal of Fluid Mechanics 2001, vol.427, pp.167–204.

[19] R. Saurel, J.P. Cocchi, P.B. Butler, Numerical study of cavitation in the wake of a hypervelocity underwater projectile, Journal of Propulsion and Power 1999, vol.15(4).

[20] O. Le Métayer, J. Massoni, R. Saurel, Elaboration des lois d'etat d'un liquide et de sa vapeur pour les modèles d'ecoulements diphasiques, International Journal of Thermal Sciences 2004, vol.43(3), pp.265–276.

[21] L. Cueto-Felgueroso, I. Colominas, X. Nogueira, F. Navarrina, M. Casteleiro, Finite volume solvers and Moving Least-Squares approximations for the compressible Navier–Stokes equations on unstructured grids, Computer Methods in Applied Mechanics and Engineering 2007, vol.(196), pp.4712–4736.

[22] S. Khelladi, X. Nogueira, F. Bakir, I. Colominas, Toward a higher order unsteady finite volume solver based on reproducing kernel methods, Computer Methods in Applied Mechanics and Engineering 2011, vol.200, pp.2348–2362.

[23] X. Nogueira, S. Khelladi, I. Colominas, L. Cueto-Felgueroso, High-Resolution Finite Volume Methods on Unstructured Grids for Turbulence and Aeroacoustics, Archives of Computational Methods in Engineering 2011, vol.18(3), pp.315–340.

[24] V. Venkatakrishnan, On the accuracy of limiters and convergence to steady state solutions 1993, 31st AIAA Aerospace Sciences Meeting & Exhibition, Reno, USA.

[25] X. Nogueira, L. Cueto-Felgueroso, I. Colominas, F. Navarrina, M. Casteleiro, A new shock-capturing technique based on moving least squares for higher-order numerical schemes on unstructured grids 2010, Computer Methods in Applied Mechanics and Engineering, vol.199, pp.2544–2558.

[26] N. Dittakavi, A. Chunekar, S. Frankel, Large Eddy Simulation of Turbulent-Cavitation Interactions in a Venturi Nozzle 2010, Journal of Fluids Engineering, vol.132.

[27] E. Shima, K. Kitamura, On new simple low-dissipation scheme of AUSM-family for all speeds 2009, AIAA paper.

[28] B. Maury, A time-stepping scheme for inelastic collisions 2006, Numerische Mathematik, vol.102(4), pp.649–679.

[29] A. Lefebvre-Lepot, Modélisation numérique d'écoulements fluide/particules, PhD Thesis, 2007.

[30] A. Merle, D. Legendre, J. Magnaudet, Forces on a high-Reynolds-number spherical bubble in a turbulent flow 2005, Journal of Fluid Mechanics, vol.532, pp.53–62.

[31] J. Magnaudet, D. Legendre, Some aspects of the lift force on a spherical bubble 1998, In Fascination of Fluid Dynamics, pp.441–461.

[32] B. Maury, Numerical analysis of a finite element/volume penalty method 2009, SIAM Journal on Numerical Analysis, vol.47(2), pp.1126–1148.

[33] Y. Wang, C.E. Brennen, Numerical computation of shock waves in a spherical cloud of cavitation bubbles 1999, Journal of Fluids Engineering, vol.121, pp.872–880.

[34] P. Tomov, S. Khelladi, C. Sarraf, F. Bakir, Numerical Modeling of Aerated Cavitation using a Penalization Approach for Air Bubble Modeling Coupled to Homogeneous Equilibrium Model, ASME IMECE 2014, Montreal, Canada.