An large eddy simulation model for simulation of turbulent flow in a converging-diverging nozzle

Aicha ABBASSI[a] and Ridha ZGOLLI[b]

a. Laboratory of Applied Mechanical Research and Engineering (LR-MAI), National School of Engineers of Tunis(ENIT),ELMANAR University, 1002 Tunis,aicha.abbassi@enit.utm.tn
b. Laboratory of Hydraulic and Environmental Modeling (LMHE), National School of Engineers of Tunis(ENIT), ELMANAR University, 1002 Tunis, ridha.zgolli@enit.utm.tn

Abstract :

Cavitation is a phenomenon with positive and negative effects, which may occur in a wide variety of applications. It is usually caused by a pressure drop accompanied by a local phase change, and, in most cases, by noise and erosion of metal walls. For instance, cavitation occurring in turbomachinery, turbopumps in rocket propulsion systems, venturi profile, hydrofoils, fuel injectors, marine propellers, nozzles are sample undesired phenomenon.
In this paper, we are interested in studying the cavitation's phenomenon in a venturi profile using OpenFOAM platform. The numerical method presented is applied to turbulent cavitating flow problems a venturi using LES. The results of the simulation of an unstable two-dimensional fluid flow in the venturi $8°$ *are presented. The results obtained from the numerical simulation are in good agreement with the experimental data. The analysis of numerical simulation data demonstrates the possibility of using the proposed method to calculate unsteady flows in venturi.*

Keywords : numerical simulation, mathematical modeling, unstable flows, Large Eddy Simulation, Re–entering jet, OpenFOAM

1 Introduction

Cavitation is a phase change phenomenon in which the water in the liquid state changes to vapor state by a decrease in pressure without the addition of heat. In fluid flows, cavitation is a multiphasic aero hydrodynamic phenomenon caused by a decrease in local static pressure in the flows, usually associated with an increase in local velocity. Thus, when the flow reaches a sufficient local velocity, the static pressure decreases to the value of the vaporization pressure and the fluid vaporizes. Therefore, the monophasic flow turns into a two-phase flow. This phase changes the produces mass, momentum, and energy transfers between the liquid and vapor phases. Physically, the negative impacts of cavitation are mainly due to the unexpected disappearance of the vapor bubbles when they meet a zone of the flow where the static pressure is higher than the vaporization pressure. In fact, in these regions, the vapor bubbles implode and generate a micro-jet that damages the materials constituting the walls solids that contain the flow.

At present, the prediction and control of cavitation is not yet assured. All industrial realities demonstrate a need for research on cavitation to better understand the mechanisms of appearance and disappearance of this phenomenon and thus provide tools and means capable of predicting the behavior of cavitating flows. These tools will then be used to improve equipment design and control of cavitating flows. For the best of our knowledge, the prediction and control of cavitation is not yet assured. All the industrial demonstrate the need for further studies on cavitation to better understand the mechanisms of appearance and disappearance of this phenomenon. Thus, these studies allow to provide tools and means predicting the behavior of cavitating flows. These tools will then be used to improve equipment design and control of cavitating flows. Cavitating flow is a complex flow to be studied experimentally and numerically. This complexity is the result of the presence of both liquid and vapor phases in the flow and thus joins the modeling problems of turbulent multiphase flow with phase change.

Studying the cavitating flows by CFD numerical simulations is also interesting for many reasons. The challenge is the ability to correctly predict the onset of cavitation and reproduce the unsteady effects of the flow. This is both for the purpose of prediction linked to industrial problems and also for accessing fine information that is difficult to measure experimentally. It is very important for manufacturers to predict cavitation development zones and implement technical solutions to confine and / or control the development of cavitation in hydraulic systems. From an industrial point of view, simulations digital devices therefore have an application interest. They have also fundamental interest because they allow not only the global analysis of the phenomenon by the prediction of unsteady behavior, but also a fine local study by analyzing the different physical parameters at different scales.

To achieve the aforementioned objectives, cavitation modeling must be based on algorithms coupling fluid dynamics equations such as Navier-Stokes equations and a cavitation model, often empirical, which must correctly predict how the vapor phase appears, disappears, and interacts with the liquid phase in a vaporization and condensation process. Cavitation models therefore control the appearance and disappearance of the vapor in the liquid flow. They often rely on the homogeneous approach defined previously : the two liquid and vapor phases constitute a diphasic mixture represented by a single fluid defined by the average physical properties.

Venturi-type section, whose convergent /divergent angle is $18°/8°$, respectively. In both cases, cavitation occurs at the Venturi throat. In the literature, there are experimental tests that have been carried out for case venturi nozzle geometry sections [1], [2] to understand the physics of the cavitating flows. On the other side, cavitation models have not changed fundamentally in recent years, both because of the lack of possibility of fine validation of the results and also because of the interaction with turbulence models. Existing turbulence models generally appear poorly adapted to the physics of cavitation and thus, in combination with cavitation models, produce deviations from reality.

In general, existing turbulence models did not take into account some important parameters such as high compressibility in mixing zones. A major difficulty on the numerical level is related to the specific developments of the resolution methods, due in particular to the character both highly compressible in the two-phase zones and almost incompressible in the pure liquid of the flow. Another important issue concerns the treatment and modeling of turbulence that strongly interacts with cavitation models.

Numerical simulations of partial cavitation in a venturi profile have been restricted to 2D simulation with Reynolds averaged Navier-Stokes RANS approach for turbulence modeling. Delgosha et al. [3] [4][5][6] used a re-normalization group RNG $k - \epsilon$ turbulence model coupled with a homogeneous equilibrium cavitation model or turbulent viscosity models used in large eddy simulation LES [7][8][9].

In our work, large eddy simulation LES was used to investigate the interactions turbulent-cavitation in the venturi geometry. The development of cavitation on Venturi geometries is the subject of many experimental campaigns that focus more particularly on the phenomenon of partial cavitation. On the other hand, this type of geometry is easy to instrument and understand the mechanisms related to the development of cavitation : on the other hand, the shape of the divergent allows reproducing the pressure fields existing on the blade profiles. This geometry was first studied in the laboratory of geophysical and industrial flows by Stutz and Reboud [1] [10]. They have experimentally defined different points in order to express cavitation behavior in the pocket.

The following sections describe the mathematical modeling, numerical methods, simulation setup, and LES results.

2 Physical description

2.1 Mixture assumptions

Cavitating flow is considered as a type of multiphase flow because liquid/vapour bubbles tend to move together with liquid phase as travelling cavity which grows in a low pressure region . However, these two phases have presented inter-phase change so that fractions for different phase are needed to analyze. Equations (1)-(3) are fundamental of multiphase flow when there is a change of phase :

$$\gamma = \frac{\forall_1}{\forall} \tag{1}$$

$$\rho = \gamma\rho_1 + (1-\gamma)\rho_v \tag{2}$$

$$\mu = \gamma\mu_1 + (1-\gamma)\mu_v \tag{3}$$

where $\forall$ and $\forall_1$ is the total mixture volume and the liquid volume respectively, γ is the liquid volume fraction. ρ is density of mixture fluid, μ is molecular viscosity of mixture fluid ; 1 and v subscripts for liquid and vapor phase.

2.2 Governing Equations

In the present numerical simulation, continuity and momentum equations are governing equations. Favre filtering operation is applied to the equations to obtain Eqs. (4) and (5). Note that the over-bar denotes filtered dependent variables.

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho\bar{u_j})}{\partial x_j} = 0 \tag{4}$$

$$\frac{\partial(\rho\bar{u_i})}{\partial t} + \frac{\partial(\rho\bar{u_i}\bar{u_j})}{\partial x_j} = -\frac{\partial\bar{p}}{\partial x_i} + \frac{\partial[\rho(R-G)]}{\partial x_j} \tag{5}$$

where $R = 2v\bar{S_{ij}}$, filtered viscous stress tensor, $S_{ij} = \frac{1}{2}(\frac{\partial(\rho\bar{u_i})}{\partial x_j} + \frac{\partial(\rho\bar{u_j})}{\partial x_i})$ rate of strain tensor, v and G= $u_i\bar{u}_j + \bar{u_i}\bar{u_j}$.
kinematic viscosity and subgrid stress tensor respectively. Equations (4) to (5) are used in large eddy simulation (LES).

2.3 Cavitation model

A mass transport cavitation model proposed by Kunz is implemented in OpenFOAM to simulate cavitating flows, which is based on the work by Merkle et al. [11]. This model considers m^+ and m^- as creation and destruction of liquid that is observed in :

$$\frac{\partial(\rho_1\gamma)}{\partial t} + \frac{\partial(\rho_1\gamma\bar{u_j})}{\partial x_j} = m^+ + m^- \tag{6}$$

Equation (7) describes the transportation rates in the cavitation model : The model used in the code is the model proposed by Merkle. For this model, the terms vaporization and condensation sources are expressed as follows :

$$m^+ = \frac{C_{prod}\rho_v\gamma^2(1-\gamma)}{t_\infty} \tag{7}$$

$$m^- = \frac{C_{dest}\rho_v\gamma min[0, p - p_v]}{(\frac{1}{2}\rho_1 U_\infty^2)t_\infty} \tag{8}$$

An artificial limitation is then necessary if one wants to respect these physical values. However, this limitation influences the conservation of the global mass because the terms m^+ and m^- intervene directly in the resolution of the different equations of the system (notably the equation of pressure correction and vacuum rate transport). In this study C_{prod} and C_{dest} are empirical constants $C_{prod} = 100$ and$C_{dest} = 100$.

$$t_\infty = \frac{L}{U_\infty}$$

undisturbed flow velocity, mean flow time scale, in which L is the characteristic length.

2.4 Flow conditions

Cavitation number is calculated using

$$\sigma = \frac{(p_r - p_v)}{\frac{1}{2}\rho U_\infty^2} \tag{9}$$

where pv, vapour pressure ; pr, reference pressure.
In our study the number of cavitation σ = 2.4.

3 Numerical Method

3.1 Geometry and mesh

We present in this section the numerical setup used to treat the unsteady cavitating flow. A two-dimensional configuration domain is used to capture the mechanisms of the growth, entrainment, and the cavitation cloud collapse. The time step has been fixed to 4.46 x 10-5 s based on the work of Coutier-Delgosha et al. (2003b) (Tref/200, where Tref = c/Uin and Uin is the velocity of flow at the inlet). The geometry employed in this study venturi profile with a 8° a closing angle before the throat 18° and an opening angle after the throat of 8°. For our case, the chord length is 0.1272 m and the input section is S = 4 x 5 mm2. The dimensions are similar to those used by Khlifa [12] during his thesis whose tests had made it possible to compare the numerical and experimental results of this geometry. To guarantee the accuracy of a LES performed on a configuration with walls, it is necessary to solve the structures of the inner zone of the boundary layer, which leads to meshes whose characteristic dimensions in this zone are : $\Delta x^+ = 100$, $\Delta y^+ = 1$ et $\Delta z^+ = 20$[14][15]. According to Sagaut [16] other values can be reported for well-resolved LES in wall pocket :, $\Delta min^+ = 1$ at the wall with at least three calculation points in the area $\Delta x^+ < 50$, $\Delta y^+ < 10$ et $\Delta z^+ < 12$. We are therefore interested in dimensionless quantities to

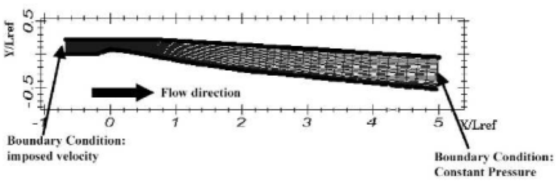

FIGURE 1 – General scheme Venturi $8°$ used by Coutier-Delgosha et al [13]

TABLE 1 – Numerical values of boundary conditions

Boundary	Condition
Inlet	Velocity in x axis U=7.8 m/s
Outlet	Pressure p=16.4 Pa
Top and Bottom	wall
Front and Back	Symmetry planes
Venturi wall	Wall

define the size of pocket mesh of the wall. These kinematic quantities are generally related to the internal parameters of the boundary layer.

$$U^+ = \frac{U}{U_\tau} \quad et \quad y^+ = \frac{y}{\delta_v}$$

With U_τ the friction velocity at the wall and $\delta_v = (v/U_\tau)$ a characteristic length of the boundary layer

The theoretical value of the first cell is computed around 1.8 x 10-6, to remain probably $y^+ = 1.510 - 6$ has been applied in the framework of mesh LES. The mesh of the venturi configuration comprises about 120,000 cells. The boundary layer is resolved to the wall. The mesh of the boundary layer is such that the size of the first mesh satisfies the condition $\Delta y^+ < 1$. Particular attention has been paid to the refinement of the mesh in the cavity area. It was applied to have a very fine discretization in the zones of the appearance of a cavitation (the throat of venturi geometry).

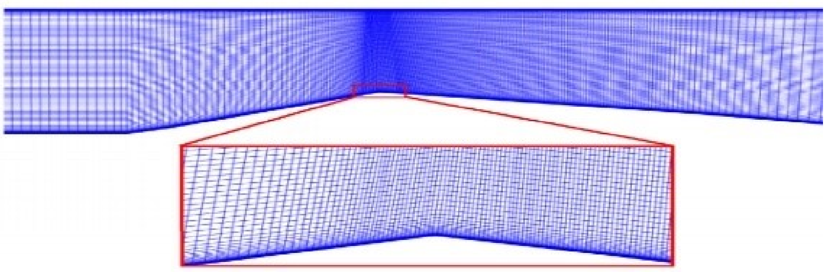

FIGURE 2 – 3D study : mesh, zoned on the section of the venturi throat

3.2 Boundary conditions

The essential question of defining the boundary conditions is : how to specify the condition at the input. In most cases, the development of downstream flow is largely dependent on the behavior of the inlet. Zhiyin [17] defined two types of conditions at the LES entry. The numerical simulation starts at 0s and lasts 2x10-1s with time steps of 1x10-4 s. The boundary conditions are described in Table 1.

4 Results

4.1 LES of sheet to could cavitation in venturi 8°

In this configuration numerical simulation has been performed on the 8° Venturi section. Comparison with the experimental data a good agreement which confirms the ability of the model to predict stable cavities. The re-entrant jet is well simulated. The experimental observation shows that the flow is unstable but there is a quasi-periodic cavitation cycle. According to the experimental conditions, the inlet velocity $U_\infty = 7.2m/s$ and the cavitation number $\sigma = 2.4$.The vapour shedding frequency is about 50Hz for a cavity length of about $45 \pm 5mm$. The pocket develops in a different way from one cycle to another and the vapor clouds are randomly dispersed in different areas of the flow. The overall evolution of the pocket

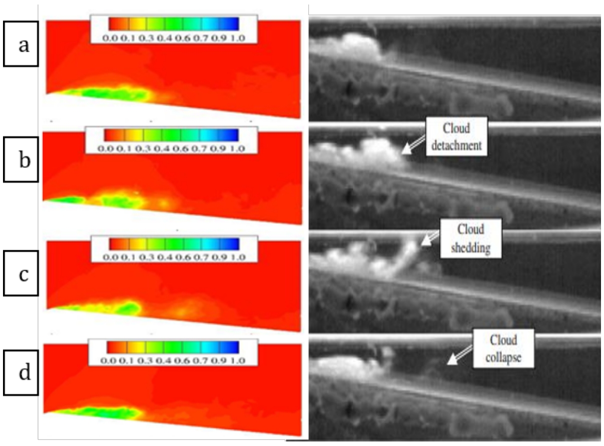

FIGURE 3 – Time series of evolution of sheet cavity in the left and experimental results in the right

follows the same cycle and their period remains almost unchanged. the evolution of the cavitation pocket for an arbitrarily chosen period : Fig 3 (a) the beginning of the cavitation cycle : The pocket begins its development upstream of the profile. In Fig 3 (b) the pocket does not remain attached to the profile and it is quickly detached and separated by a zone of liquid : this behavior suggests that the physical model of cavitation used does not make it possible to correctly model the zone two-phase diphasic. the vapor clouds from the previous cycle almost all imploded. The incoming jet rises to reach the upstream of the pocket and causes the detachment of the vapor cloud in Fig 3 (c). In Fig 3 (d) a new cavitation pocket appears at the throat.

4.2 Re-entering jet

The negative values, noticed near the lower wall on the velocity profiles, reveal the intermittent presence of the re-entrant jet which cuts off the pocket and causes its detachment. For visualizing the re-entering jet, span averaged values are considered. Although spanwise variation in the re-entering jet is present, this method will give us details about the mean behavior. Streamlines plotted at three different time instants within a cycle are shown in Fig. 4 and it shows the presence of a re-entering jet where a stream of liquid from the cavity closure enters into the cavity.

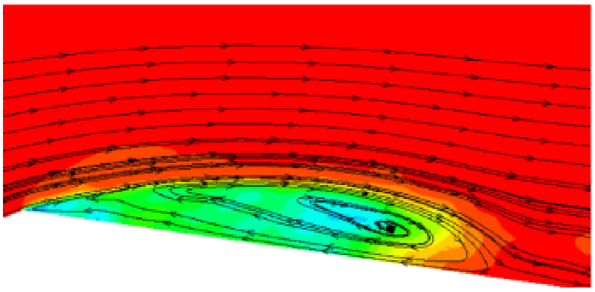

FIGURE 4 – Re-entering jet

4.3 Effects of turbulence modelling on cavitation modelling

The complex, unsteady mechanism that governs the cyclic cavitation behaviour is strongly affected by the applied turbulence model. Coutier-Delgosha et al. [18] compared some of the turbulence models, such as $k-\epsilon$, modified $k-\epsilon$, and $k-\omega$ with and without compressibility. They concluded that considering the compressibility is vital for cavity dynamics simulation, but these models are categorized as RANS models. In contrast to RANS models, LES models solve the large eddies directly and just use the model for the small-scale eddies. Indeed, with the LES models, more physics is considered and as a result more computation time is consumed. However, for the simulation of cavity dynamics, it is of primary importance to apply LES models instead of time-averaged RANS models because of the inherent unsteady turbulent structure.

5 Conclusion

A study of unsteady cavitating flows has been presented in Venturi-type sections. The experimental data show a stable cavitation sheet an unsteady behaviour of the cavitating flow in a $8°$ Venturi-type section, which were both simulated via a Merkle cavitation model and by applying LES as a turbulence model. A numerical method using characteristic-based filtering developed to simulate multiphase and cavitating flows is used to perform Large Eddy Simulation of cavitation. A homogeneous equilibrium model is used to model the multiphase mixture as a single compressible fluid. A characteristic-based filter developed for multiphase flows is applied in a predictor corrector method to make it independent of the base scheme. Reasonable agreement is obtained with the available experimental data. The method is used to perform LES of sheet to cloud cavitation in a venturi geometry. A re-entering jet of liquid is found to be responsible for the sheet to cloud transition.

Références

[1] B .Stutz, J .Reboud, Experiments on Unsteady Cavitation, Exp. Fluids (1997) 191-198

[2] Matevz. Dular, I. Khlifa, Scale effect on unsteady cloud cavitation. Experiments in fluids (2012), 1233-1250

[3] O.Coutier-Delgosha,B.Stutz,A.Vabre,S.Legoupil, Analyse of cavitating flow structure by experimental and numerical investigations.,Journal of fluid mechanics(2007),171-222

[4] O.Coutier-Delgosha,Modélisation des écoulements cavitants : Etude des comportements instationnaires et application tridimensionnelle aux turbomachines, Thèse université Grenoble INPG, 2001

[5] O .Coutier-Delgosha, J.F.Devillers, T.Pichon, A.Vabre, R.Woo, S.Legoupil, Internal structure and dynamics of sheet cavitation,Physics of Fluids (2006) p.017103

[6] O. Coutier-Delgosha, J. L.Reboud, Y.Delannoy, Numerical simulation of the unsteady behaviour of cavitating flows. International Journal for Numerical Methods in Fluids (2003) 527 ?548

[7] E .Roohi, A.P. Zahiri, M .Passandideh-Fard, Numerical simulation of cavitation around a two-dimensional hydrofoil using VOF method and LES turbulence model. Applied Mathematical Modelling (2013),6469-6488

[8] X .Yu, C.Huang , T. Du, L. Liao, X. Wu , Z .Zheng, Y.Wang,Study of Characteristics of Cloud Cavity Around Axisymmetric Projectile by Large Eddy Simulation. Journal of Fluids Engineering (2014) p 051303

[9] N. M. NOURI, S. M. H. MIRSAEEDI, M. MOGHIMI, Large eddy simulation of natural cavitating flows in Venturi-type sections. Journal of Mechanical Engineering Science (2010) 369-381

[10] B.Stutz, J.L .Reboud, Two-phase flow structure of sheet cavitation. Physics of Fluids (1997), 3678-3686.

[11] C. L.Merkl., , J. Z.Feng, P. E. O.Buelow, Computational modeling of the dynamics of sheet cavitation. In Proceedings of the 3rd International Symposium on Cavitation,(1998), 307 ?311

[12] I. Khlifa, O. Coutier-Delgosha, M. Hocevar, S. Fuzier, A. Vabre, K. Fezzaa et W. Lee, Fast X-Ray imaging for velocity measurements in cavitating flows,Cav2012 (2012)

[13] O. Coutier-Delgosha,R.Fortes-Patella,J.L.Reboud,B.Stutz, Unsteady cavitation in a venturi type section,Mutiphase Science and technology(2004),205-215

[14] U.Piomeli, Wall layer models for large eddy simulations, Progress in Aerospace Sciences(2008),437-446

[15] U.Piomeli,Large eddy simulation :archievements and challenges, Progress in Aerospace Sciences(1999),335-362

[16] P.Saugaut,S.Deck,M.Terracol,Mutiscale and multiresolution approaches in turbulence,chapter8 :Zonal RANS/LES Methodes,Imperial college Press (2006)

[17] Y.Zhiyin,Large eddy simulation :Past, present and the future, Chinese Journal of Aeronautics(2015),11-24

[18] O.Coutier-Delgosha, , R.Fortes-Patella, J. L.Reboud, Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation. Journal of Fluids Engineering (2002), 38 ?45