STUDY OF PARTIAL CAVITATION ON A PLANE-CONVEX HYDROFOIL WITH MESH DEVELOPMENT BY USING GMSH FREE SOFTWARE

Victor Hidalgo
State Key Laboratory of Hydro science & Engineering, Tsinghua University, Beijing 100084, China
victor.hidalgo@epn.edu.ec

XianWu Luo *
State Key Laboratory of Hydro science & Engineering, Tsinghua University, Beijing 100084, China
luoxw@mail.tsinghua.edu.cn

Xavier Escaler
Centre for Industrial Diagnostics & Fluid Dynamics, UPC, Barcelona, Spain

An Yu
State Key Laboratory of Hydro science & Engineering, Tsinghua University, Beijing 100084, China

Esteban Valencia
Department of Mechanical Engineering, Escuela Politecnica Nacional, Quito, E11-253, Ecuador

ABSTRACT

Commercial programs are widely used to do unstructured and structured meshes for CFD simulations. However, grids and meshes based on free-open source software (FOSS) give to researchers and engineers the possibility to adapt and improve the meshing process for special study cases with a high Reynolds numbers, such as unsteady partial cavitating flows. In order to improve the grid qualities, the FOSS GMSH has been used to do three types of grid, unstructured hexahedral mesh, hybrid mesh and structured hexahedral mesh for the simulation of partial cavitation around a plane-convex hydrofoil. Numerical simulations have been carried out by using the FOSS OpenFOAM based on the Zwart cavitation model and the implicit large eddy simulation (ILES). The results show that the structured mesh provides the best simulating to experimental data. On the other hand, the hybrid mesh induces unreliable results at leading edge without shedding.

NOMENCLATURE

AOA Angle of attack.

L, V Subscripts for liquid and vapor.
i, j Subscripts of axes 1, 2 or 3.
+ Superscript for evaporation.
− Superscript for condensation.
∀ Total volume.
m Mass transfer rates per volume.
R B Typical bubble size in water.
u Variable of velocity.
p Variable of pressure.
t Variable of time.
x Variable of space.
L e Edge length in the mesh.
∀D Volume of a hexahedron in a mesh.
∑k A k Sum of the areas of a hexahedron in a mesh.
pr Reference pressure downstream.
U ∞ Main flow stream velocity.
α Fraction of vapor volume.
ρ Density of total fluid.
μ Dynamic viscosity of total fluid.
v Kinematic viscosity.
τ ij Filtered viscous stress tensor 2νSij.
\( \bar{S}_{ij} \) Strain rate tensor \( \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \).

\( \tau'_{ij} \) The subgrid scale tensor \( \left( \mu \rho \left( u_i u_j - \bar{u}_i \bar{u}_j \right) \right) \).

\( \sigma \) Cavitation number.

\( \xi \) Dimensionless time for cavitation cycle.

INTRODUCTION

Computational fluid dynamics (CFD) is shown as an important tool in the study of the partial cavitation, due to costs and limitations of experiments [1, 2]. In this way, several cavitation models based on mixture homogeneous flows have been presented to predict the cavity shedding and pressure fluctuation [3, 4]. Hidalgo et al [5–7] have tested three cavitation models by using OpenFOAM. The results proved that Zwart model approaches the experiments.

The Reynolds Average Navier Stokes method, RANS, used to solve the momentum and continuity equations does not capture accurately the unsteady partial cavitation phenomena. In order to improve the quality of the simulation for unsteady cavitating flows with high Reynolds numbers, correction parameters must be included in the main equations [8–12]. The Large Eddy Simulation method, LES, intends to resolve this problem. Large unsteadies motions of the flow field are estimated and only small scales are modeled by a subgrid scale model, SGS, in explicit methods of LES [9, 13–15]. ELES. The SGS is avoided in the implicit method of LES, ILES, by using the truncation error to act as a dissipative action [16–18] for correcting excessive dissipation of ELES.

For these reasons, ILES and Zwart model have been used in this study for the numerical simulation of partial cavitation in a plane convex hydrofoil. However, the discussion about mesh generation as an "art or science" from Timothy et al. [19] shows that the mesh is considered as one of the main parts in CFD. It has a significant role in the outcome of numerical simulations, because a poor mesh could induce wrong results [20, 21]. In this context, the research challenge in this study is to conclude the influence of mesh type on a 2D case of partial cavitation.

PHYSICAL DESCRIPTION

The partial cavitation phenomena is modeled as a mixture homogeneous flow [5], so that, the vapor volume fraction, \( \alpha \), is introduced as indicated in Eqn.1 to calculate density, \( \rho \), and dynamic viscosity, \( \mu \) with Eqn.2 and Eqn.3.

\[
\alpha = \frac{\nu}{\bar{V}}; \quad (1)
\]

\[
\rho = \alpha \rho_v + (1 - \alpha) \rho_L; \quad (2)
\]

\[
\mu = \alpha \mu_v + (1 - \alpha) \mu_L; \quad (3)
\]

ILES method

The filtered equations of continuity and momentum [5] are used in ILES and they are shown in Eqn.4 and Eqn.5.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0; \quad (4)
\]

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial \left( \tau'_{ij} - \tau'_{ij} \right)}{\partial x_j}; \quad (5)
\]

To introduce \( \alpha \) in Eqn.4, the transport equation is obtained, which is indicated in Eqn.6.

\[
\frac{\partial (\alpha \rho_v)}{\partial t} + \frac{\partial (\alpha \rho_v \bar{u}_i)}{\partial x_i} = \bar{m}; \quad (6)
\]

Finally, the no-homogeneous expression for velocity divergence is written as indicated in Eqn.7.

\[
\frac{\partial \bar{u}_i}{\partial x_i} = \bar{m} \left( \frac{1}{\rho_v} - \frac{1}{\rho_L} \right); \quad (7)
\]

Zwart Cavitation Model

The Zwart cavitation model (Eqn.8) based on Rayleigh-Plesset equations was developed by using C++ and implemented in OpenFOAM [7], because this is not included in the library of solvers.

\[
\bar{m} = \begin{cases} 
F_V \frac{3 r_{nuc} (1 - \alpha) \rho_v}{R_B} \sqrt{\frac{2}{3}} \left( \frac{p_v - p}{\rho_L} \right) & \text{if } p < p_v \\
- F_C \frac{3 \alpha \rho_v}{R_B} \sqrt{\frac{2}{3}} \left( \frac{p - p_v}{\rho_L} \right) & \text{if } p > p_v
\end{cases}; \quad (8)
\]

where \( F_V = 300 \) and \( F_C = 0.03 \) are the selected calibration constants for vaporization and condensation [22], \( r_{nuc} = 5.0 \times 10^{-6} \) is the nucleation site volume fraction and \( R_B = 1.9 \times 10^{-6} \) [m] is the typical bubble size in water [2].
The plane convex hydrofoil used for cavitation erosion tests at École Polytechnique Fédérale de Lausanne (EPFL) [23] was considered. The hydrofoil is shown in Fig. 1, with 3° of AOA. A semicircular leading edge and a tip trailing edge connect the upper and lower surfaces.

The computation domain and boundary conditions are shown in Fig. 2 with \( U_\infty = 25 \text{ [m/s]} \) at the inlet, \( p_r = 2.2 \text{ [bar]} \) at the outlet, and symmetry planes for front and back faces are set up as . It is noted from Fig. 2 that the study has been considered as a 2D case, due to the spanwise length, which is equal to 0.016 of \( c \).

### Cavitation Condition

The cavitation condition is \( \sigma = 0.7 \), which was predicted with Eqn. 9.

\[
\sigma = \frac{p_r - p_V}{\frac{1}{2} \rho U_\infty^2}
\]  

### MESH

GMSH has been used for meshing the domain. It is an free software distributed under the terms of GNU Generic Public License (GPL), entirely written in standard C++ with a friendly interface [24]. The reasons behind its selection are: i) enables the use of geometries in .geo, ii) exports to a variety of formats, iii) imports different CAD and mesh for optimizing. The .msh result is transformed to OpenFOAM mesh with the command gmshToFoam.

Three types of mesh are developed based on previous studies [21] as shown in Fig. 3. The unstructured mesh (a) has 27118 hexahedra, the hybrid mesh (b) 19737 hexahedra and the structured mesh (c) 20011 hexahedra. Equations 10 and 11 have been used to evaluate the quality of mesh. They are based on recommendation from Christophe Geuzaine and Jean-François.
gamma = \frac{\forall D}{\sum_k A_k \times \max(L_e)}; \quad (10)

rho = \frac{\min(L_e)}{\max(L_e)}. \quad (11)

The gamma analysis is related to the volume of an hexahedron element in the mesh, \forall D, and the rho analysis is related to the size of elements based on the edge length, \(L_e\), for an adequate distribution of elements.

**SOLVER**

The OpenFOAM version 2.2.x has been used as a CFD software based on previous studies of unsteady cavitating flows around a NACA0015 and NACA 66 [5–7]. The Zwart Cavitation Model was compiled to get a new solver called vInterPhaseChangeFoam.

**RESULTS AND DISCUSSIONS**

The gamma analysis for mesh quality is shown in Fig. 4, where values near 1 are related to the more appropriate hexahedron element for ILES calculation. The unstructured hexahedral mesh (a) shows semi-aleatory distribution of values from 0.5 to 1. The hybrid mesh (b) in the structured part, which is used to catch the cavitation phenomenon, shows four regions with values close to 1. They are located at the leading edge, upper and lower center of the hydrofoil and trailing edge. Other four regions with values from 0.5 to 0.7 are located between the first described ones. The structured hexahedral mesh (c) shows very clearly a homogeneous region with a value equal to 1 in the calculation zone. Thought the zones near to the trailing edge and before the semicircular edge present values from 0.5 to 1.

Plots of elements number as a function of rho for cases (a), (b) and (c) are shown in Fig. 5. Case (a) presents two peaks more than the others between 0 and 0.25. The maximum pulse is \(1.14 \times 10^3\) elements, which matches the case (b), but not (c), which is \(1.09 \times 10^3\). Though the case (b) drops faster than (a) and (c) before 0.25 rho, the number of elements for case (b) remains between 288 and 144 before 0.75, and after the number of elements drops to zero. The case (a) from 0.25 to 1.0 rho presents two small peaks equal to 343 and 228 elements, and the case (c) shows a gradual behavior with a significant peak equal to 326 elements between 0.25 and 0.5.

In this context, for better understanding the rho quality, the hv analysis is introduced, which is based on Eqn. 12. hv refers to the relation between the maximum value of the number of elements from rho analysis, \(N_{\rho o}\), and the total number of elements from the mesh, \(N_m\).
Calculated $h_v$ values for cases (a), (b) and (c) are $4.2 \times 10^{-2}$, $5.8 \times 10^{-2}$ and $5.4 \times 10^{-2}$ respectively. This value indicates that the case (c) is in the middle.

Figure 6 shows the comparison between the upper surface view of the experimental results from EPFL and the transversal surface plots of numerical results for cases (a), (b) and (c). All of the cases show similar to the experimental data. The lengths of the attached cavity sheets for (a) and (b) are shorter than (b) and more similar cavitation to experimental result at 0.5 of the cavitation cycle, $\Delta t$. However, the reentrant jet, the shape of the cavity sheet, the cloud of bubbles and the vapor quantity are better simulated in (c). It is noted that for (b) the mesh induces some results at the lower surface near to the leading edge, which are not reliable.

CONCLUSIONS

In this research the influence of mesh type on the simulation of partial cavitation for a plane-convex hydrofoil was analyzed. In this context the following milestones are concluded:

1. The mesh obtained by GMSH free software presents good qualities to simulate partial cavitation, based on gamma and rho analysis. The gamma analysis shows that the structured hexahedral mesh presents zones with values near to 1, which are better for ILES. A good form of elements distribution is presented by rho. Based on this analysis the structured mesh presents the best qualities of distribution and an intermediate $h_v$.

2. ILES method and Zwart Cavitation Model developed and implemented in OpenFOAM 2.2.x show that the solver
could hand different type of meshes and reproduce the experimental observations.

ACKNOWLEDGMENT
This work was financially supported by the National Natural Science Foundation of China (Project Nos. 51306018, 51206087 and 51179091) and State key Laboratory of Hydroscience and Engineering, Tsinghua University (Project Nos. 2014-KY-05 and 2015-E-03).

REFERENCES