Benefits of hydropower research in Ecuador using OpenFOAM based on CFD technology

A practical cavitation study for NACA0015

Víctor Hugo Hidalgo D.
State Key Laboratory of Hydroscience & Engineering, Tsinghua University
Beijing 100084, China
Email: victor.hidalgo@epn.edu.ec

XianWu Luo
State Key Laboratory of Hydroscience & Engineering, Tsinghua University
Beijing 100084, China
Email: luoxw@mail.tsinghua.edu.cn

Adrián Patricio Peña
Department of Mechanical Engineering, Escuela Politécnica Nacional
Quito E11-253, Ecuador
Email: patricio.pena@epn.edu.ec

Esteban Alejandro Valencia T.
School Of Engineering, Cranfield University
Cranfield Bedfordshire, United Kingdom
Email: e.valencia@cranfield.ac.uk

Ricardo Soto
Department of Mechanical Engineering, Escuela Politécnica Nacional
Quito E11-253, Ecuador
Email: ricardo.soto@epn.edu.ec

An Yu
State Key Laboratory of Hydroscience & Engineering, Tsinghua University
Beijing 100084, China
Email: yua12@mails.tsinghua.edu.cn

***Abstract*—Large eddy simulation model (LES) and the open source OpenFOAM are used in the present numerical study of unsteady cavitating flow around a NACA0015. The results showed a great accuracy with experiments. The validations of the present simulations are strong advantage to solve cavitation problems of turbines for hydroelectric plants which represents more than 46% of electricity in Ecuador.**

***Keywords*—Cavitation; LES; Open-source; OpenFOAM; python; turbine.**

I. INTRODUCTION

Electrical energy generation based on hydromechanics presents numerous advantages for countries as Ecuador, where high caudal water sources are available. In fact todays' energy production represents 46.52% (Fig. 1), which is aimed to increase up to 70% in the coming years. For this purpose, the Ecuadorian government has impulsed the creation of new hydroelectric projects.

Coca codo sinclair is one of this new projects, which is constructed in collaboration with China and will represent 1500MW of energy production [1], [2].

Although hydroelectrics represent a feasible and optimum way of producing electricity, because of its high technology readiness level and low environmental impact. It also comes with design challenges. Erosion of turbine blades caused by cavitation is certainly one of these, as it contributes to reduce cycle efficiency and turbine lifting . This aspect motivates to develop a method to assess this phenomena and provide recommendations for the system design. In this scenario, countries as Ecuador which depend by large extent on hydropower can be greatly benefited. This work focuses on the study of cavitation phenomena using a through flow methods (CFD). For this opensource platforms have been used.

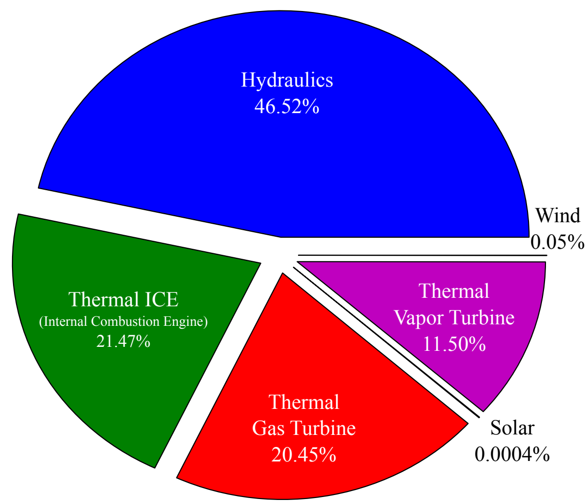

Figure 1: Distribution of electricity generation by power plant in percentage [1]

In order to understand the physyics of cavitation and erosion. These phenomena are describe in the following paragraphs.

Cavitation is a physical phenomena, which occurs when there is a phase change from liquid to vapor due to pressure variations.

Erosion is a result of unsteady cavitating flow. The phenomenon starts when a cavity sheet appears over a solid surface because of unsteady flow and transforms to a group of bubbles, some of these bubbles collapse and pressure waves are emitted, which reach other other bubbles close to the surface and produce oscillation. Therefore, a micro-jet is generated with hight velocity, causing a single pit(erosiondamage) over the surface [3].

In fact, developing of predicted cavitating flow by computational fluid dynamics model help to improve designs in hydromachines as turbines and pumps.

Kubota in 1992 gave to the science community a model based on homogeneous flow whit mixture density, considering a single fluid for Navier-Stokes equations, using Rayleigh's equation and local void fraction [4]. Basing the model on transport equation and volume of liquid or vapor fraction are presented for numerical investigation [5], [6], [7], [8].

The unsteady cavitating flow most of the time shows hight Reynolds number, so that, Reynolds Average Navier Stoke (RANS) presents some problems for solving the cavity growth. Therefore, some adaptations are necessaries to get better results [9], [10], [11]. Large eddy simulation (LES) are developed to solve this problem and results are showing good precision and accuracy to predict large scale turbulent eddies [12].

Understanding and predicting this physical phenomenon by CFD simulations are the aims for the present research. There are several privative software as Ansys Fluent, CFX, COMSOL and others, which could be used to perform numerical studies. However, Free - Open sources software as OpenFOAM, SALOME and python are the las challenge, because open code permits to do any change by any person and improving the solver of CFD [13].

II. Physical Model

A. Mathematical Assumptions

Basing the cavitation on a multiphase flow, is conformed part with a liquid fraction and vapor fraction (1).

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

where $\forall$ represent the volume of fluid and γ is the liquid volume fraction. Therefore, density and viscosity based on this idea are observed from (2) and (3).

$$\rho = \gamma\rho_L + (1-\gamma)\rho_V \tag{2}$$

$$\mu = \gamma\mu_L + (1-\gamma)\mu_V \tag{3}$$

where L and V represent liquid and vapor, ρ is the total density of fluid and μ is the total viscosity.

B. Cavitation number

The cavitation number σ is calculated by (4) to define unsteady flow conditions, where p_r is the absolute pressure of medium,p_V is the saturation pressure and U_∞ is a velocity of flow inside the simulation.

$$\sigma = \frac{p_r - p_V}{\frac{1}{2}\rho U_\infty^2} \tag{4}$$

C. Equations

Favre-filtering operation is applied to continuity and momentum equations for LES, as observed from (5) to (7).

$$\frac{\partial \overline{u}_j}{\partial x_j} = \dot{m}\left(\frac{1}{\rho_L} - \frac{1}{\rho_V}\right) \tag{5}$$

$$\frac{\partial \gamma}{\partial t} + \frac{\partial(\gamma \overline{u}_j)}{\partial x_j} = \frac{\dot{m}}{\rho_L} \tag{6}$$

$$\frac{\partial(\rho\overline{u}_i)}{\partial t} + \frac{\partial(\rho\overline{u}_i\overline{u}_j)}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\rho(R_a - G)\right] \tag{7}$$

where:

i, j	are the axis direction 1, 2 or 3 in the space
$\overline{u}$	is the filtered velocity for a particle of fluid
$\overline{p}$	is the filtered pressure
x	is the space axis 1, 2, or 3
t	is the time for the physical phenomenon
$\dot{m}$	is the mass rate in $kg/(m^3 s)$
Ra	is $2\nu\overline{S}_{ij}$ filtered viscous stress tensor
$\overline{S}_{ij}$	is the rate strain tensor
ν	is the kinematic viscosity, and
G	is the subgrid stress tensor.

D. Cavitation Model

The model of Schnerr-Sauer is used in the present study. The model is based on Rayleigh-Plesste equation, as explained in section I and indicated in (8).

$$R\frac{\partial^2 R}{\partial t^2} + \frac{3}{2}\left(\frac{\partial R}{\partial t}\right)^2 = \frac{p - p_V}{\rho_L} \tag{8}$$

where R is the a radius of bubble. Equation (9) shows that second derivate of (8) is neglected for the math model.

$$\frac{dR}{dt} = \sqrt{\frac{2}{3}\frac{|p_V - p|}{\rho_L}} \tag{9}$$

In Schnerr-Sauer model is used α as vapor volume fraction, considering that $1 = \alpha + \gamma$. R is calculated by using a density of bubbles per volume $\forall$ in (10) and (11). However, (9) could be used to find R.

$$\alpha = \frac{n\frac{4}{3}\pi R^3}{1 + n\frac{4}{3}\pi R^3} \tag{10}$$

$$R = \sqrt[3]{\frac{\alpha}{\frac{4}{3}\pi n(1-\alpha)}} \tag{11}$$

Equation (12) is the final part to calculate the mass transfer, where C_e and C_c basing their values on the process of condensation and vaporization [14], the coefficients are 2 and 1 respectively, $\dot{m}^+$ and $\dot{m}^-$ are the rate mass creation and destruction of vaporization-condensation.

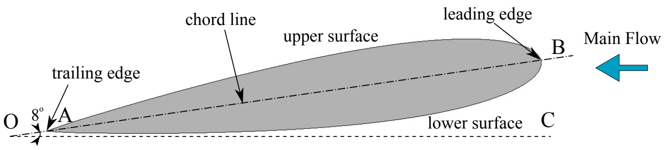

Figure 2: Constitutive parts of NACA-0015

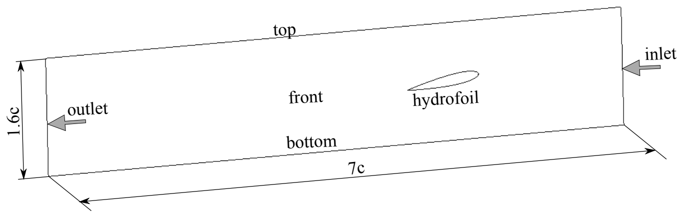

Figure 3: Domain of numerical study

TABLE I: Components of Structured Mesh

Data type	Number
Nodes	50600
Quadrangles	51370
Hexahedrons	24920

$$\begin{cases} \dot{m}^{+} = C_e \dfrac{\rho_V \rho_L}{\rho} \alpha(1-\alpha) \dfrac{3}{R} \sqrt{\dfrac{2(p_V - p)}{3\rho_L}} \\ \dot{m}^{-} = C_c \dfrac{\rho_V \rho_L}{\rho} \alpha(1-\alpha) \dfrac{3}{R} \sqrt{\dfrac{2(p - p_V)}{3\rho_L}} \end{cases} \tag{12}$$

III. The Geometry Profile

A. Shape

Fig. 2 shows a NACA-0015 used in the research, where the distance $\overline{AB}$ is the chord line c, the angles $\widehat{BOC}$ is 8^o and is the attack angle.

The domain of numerical simulation is observed in Fig. 3.

B. Structured Mesh

The quality of mesh is important in LES [12], [15], so that a structured mesh is selected for meshing as shown in Fig. 4.

The flow around the hydrofoil wall is captured by a special scale distribution mesh as shown in Fig. 4, this mesh is drawn by Salome 7.2.0 (Open Source) and validated by YPlus (y^+) number (13), where u_τ is the friction velocity close to hydrofoil wall, y is the distance from nearest hydrofoil wall, and μ is the kinematic viscosity.

$$y^{+} = \frac{u_\tau y}{\nu} \tag{13}$$

Table I gives information about components of the structured mesh.

IV. Solver

A. OpenFOAM

Open source Field Operation and Manipulation (OpenFOAM) are libraries of C++ to develop numerical solvers

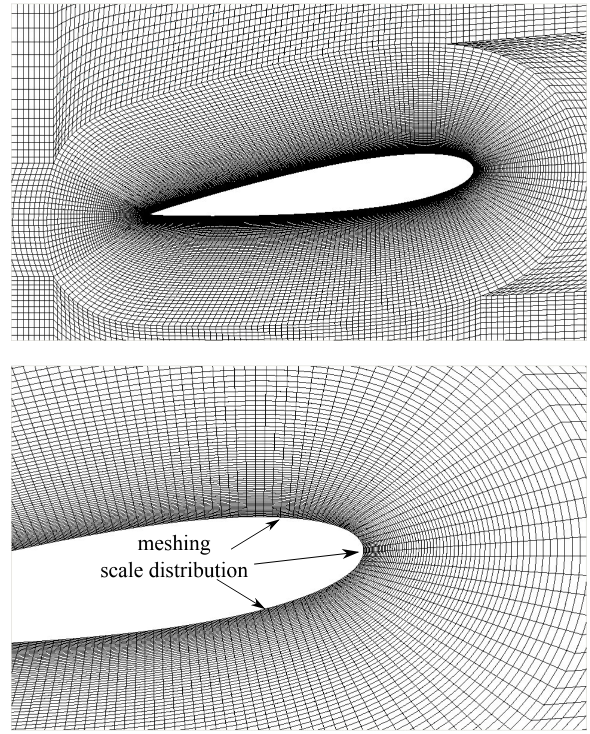

Figure 4: Structured mesh of NACA-0015

TABLE II: Boundary Conditions of NACA-0015

Inlet	velocity in x axis $U_\infty = 5.477$ m/s
Outlet	pressure $p = p_r = 20.3$ kPa
Top and Bottom	wall
Front and Back	symmetry planes
Saturated vapor pressure	p_V = 2.3 kPa

for CFD [16]. In fact, Schnerr-Sauer cavitation model is a default solver in OpenFOAM 2.1.1 to study flow with interphase change like vapor-liquid water. The given value of n for R is 10^7 .

B. Setup

Table II is the data for setup conditions of numerical simulation. The time start to $0s$ and end to 6.5×10^{-1} with a delta time of 2×10^{-5} and write result every 25 steps.

V. Results and Discussions

The unsteady cavitating flow appear due to the fact that the calculated σ is 1.2, and the y^+ obtained is less than 1 as “in press” [17].

A specific time ξ is used based on (14) to get Fig. 5, where initial and final time are represented by t_o and t_f.

$$\xi = \frac{t - t_o}{t_f - t_o} \tag{14}$$

Fig. 5 shows how the pressure change for a typical cavity sheet cycle in 0.2 of x/c, which is a point located near leading edge. In Fig. 5 is observed two limits that represents p_r and p_V, if the pressure is close to or lower than p_V means that

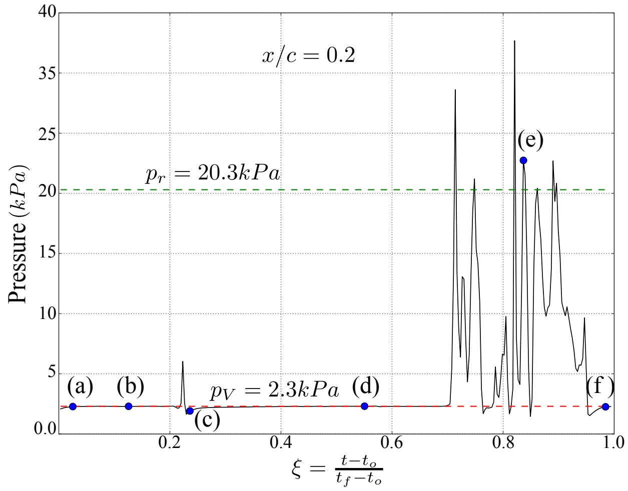

Figure 5: Pressure fluctuation for a break-off cycle in $x/c = 0.2$

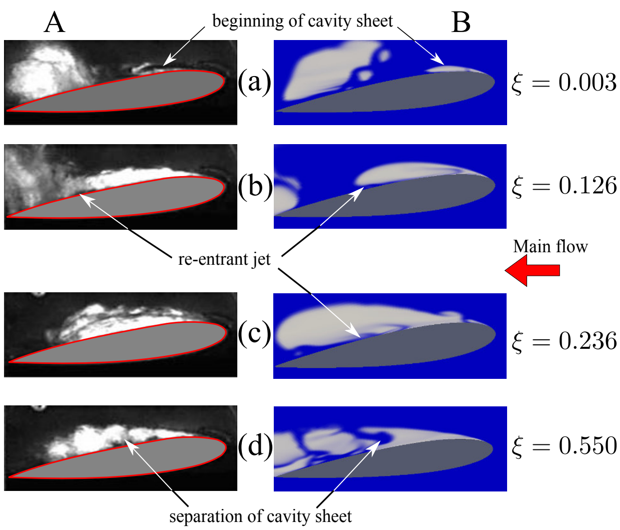

Figure 6: Comparisons between experiment and simulation result of cavity sheet cycle

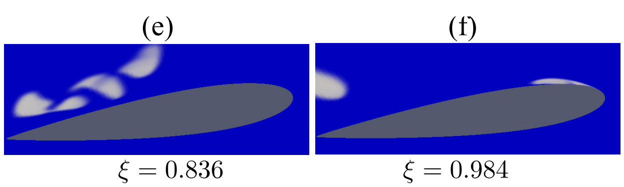

Figure 7: Bubble motion and restart of break-off cycle

there is change of phase to vapor and when the pressure is raised, the vapor phase return to liquid. In fact, the performed numerical simulation shows that pressure is near saturation from ξ equal to 0, after 0.6 and before 0.8, then, there are some irregular picks before the final of cycle.

There are remarked points from (a) to (f) in Fig. 5, which are used to obtain Fig. 6 and Fig. 7, describing a typical break-off cycle.

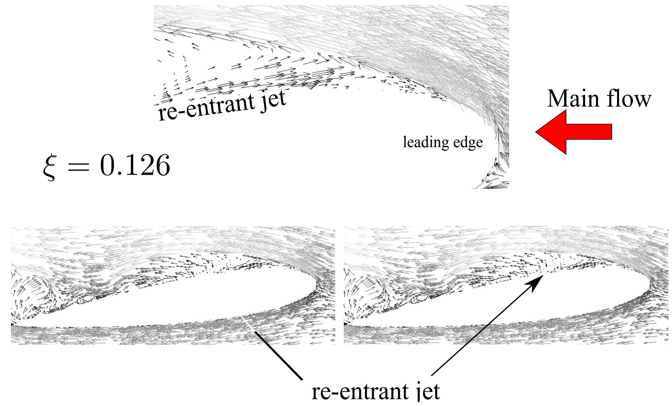

Figure 8: Velocity field and reentrant-jet effects

The results of study are compared and validated by experimental data available from research of Yakushiji R. et. al [18] as shown in Fig. 6. The column A is data from experiment and B are numerical results. The cycle start when a cavity sheet appears close to leading edge at (a), then a generated reentrant-jet moves from trailing edge to leading edge in the opposite direction of main flow at points (b)-(c), and there is a separation of a group of bubbles from the cavity sheet as shown in (d).

Fig. 7 shows the last part of break-off cycle, when the cavity sheet transform to group of bubbles with circular motion in (e) as "in press" [17], and restarting a new cycle due to the occurrence of another cavity (f).

Fig. 8 gives information about the velocity field and the effects of reentrant-jet when ξ is 0.0.126, it is observed that reentrant-jet and main flow velocities are opposite directions, so that, the cavity sheet separated from the surface of hydrofoil because reentrant-jet effects. Because the velocity direction the bubbles get a circular motion.

VI. Conclusion

The following milestones are the conclusion of this numerical research:

1) The numerical method implemented shows optimal solution to predict cavitation and cavitating flow.
2) Results shows that the use of a dimensionless number of time for a typical break-off cycle analysis is a better option to get a general model for cavitation-erosion model.
3) Great similitudes between experiments and numerical results from OpenFOAM, express that mathematical model as LES could be used to improve designs and maintenance of hydrodynamics machinery in countries like Ecuador.

Acknowledgment

This work was financially supported by the National Natural Science Foundation of China (Project Nos. 51206087 and 51376100), and the Major National Scientific Instrument and Equipment Development project (Grant No.

2011YQ07004901). The authors also would like to thank National Secretary of Science and Technology of Ecuador (SENESCYT).

REFERENCES

[1] H. L. S. Fabin and G. d. l. M. J. Ángel, "Estudio Hidroenergtico - PH Coca Codo Sinclair [Ecuador]," *XXII Congreso Nacional de Hidráulica, Acapulco, Guerrero, México*, 2012.

[2] C. L. R. Ochoa, "Criteria for pre-fesibility and environmental impact assessments of hydroelectric projects in ecuador," Master's thesis, UNIVERSITY OF CALGARY, 2002.

[3] M. Dular, B. Stoffel, and B. Sirok, "Development of a cavitation erosion model," *Wear*, vol. 261, no. 5, pp. 642 – 655, 2006.

[4] A. Kubota, H. Kato, and H. Yamaguchi, "A new modelling of cavitating flows: A numerical study of unsteady cavitation on a hydrofoil section." *Journal of fluid Mechanics*, vol. 240, no. 1, pp. 59 – 96, 1992.

[5] A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, "Mathematical basis and validation of the full cavitation model," *Journal of Fluids Engineering*, vol. 124, no. 3, pp. 617 – 624, Aug. 2002.

[6] R. F. Kunz, D. A. Boger, D. R. Stinebring, T. S. Chyczewski, J. W. Lindau, H. J. Gibeling, S. Venkateswaran, and T. R. Govindan, "A preconditioned NavierStokes method for two-phase flows with application to cavitation prediction," *Computers & Fluids*, vol. 29, no. 8, pp. 849 – 875, 2000.

[7] X. Zhang, Z. Wu, S. Xiang, and L. Qiu, "Modeling cavitation flow of cryogenic fluids with thermodynamic phase-change theory," *Chinese Science Bulletin*, vol. 58, no. 4-5, pp. 567 – 574, Feb. 2013.

[8] X. Cao, X. Zhang, L. Qiu, and Z. Gan, "Validation of full cavitation model in cryogenic fluids," *Chinese Science Bulletin*, vol. 54, no. 10, pp. 1633 – 1640, May 2009.

[9] B. HUANG and G.-y. WANG, "Partially averaged navier-stokes method for time-dependent turbulent cavitating flows," *Journal of Hydrodynamics, Ser. B*, vol. 23, no. 1, pp. 26 – 33, Feb. 2011.

[10] B. Ji, X.-W. Luo, Y.-L. Wu, and H.-Y. Xu, "Unsteady cavitating flow around a hydrofoil simulated using the partially-averaged NavierStokes model," *Chinese Physics Letters*, vol. 29, no. 7, p. 076401, 2012.

[11] B. Huang, G. Wang, Y. Zhao, and Q. Wu, "Physical and numerical investigation on transient cavitating flows," *Science China Technological Sciences*, vol. 56, no. 9, pp. 2207 – 2218, Sep. 2013.

[12] B. JI, X.-w. LUO, X.-x. PENG, and Y.-l. WU, "Three-dimensional large eddy simulation and vorticity analysis of unsteady cavitating flow around a twisted hydrofoil," *Journal of Hydrodynamics, Ser. B*, vol. 25, no. 4, pp. 510 – 519, Sep. 2013.

[13] W. Scacchi, "The future of research in free/open source software development," in *Proceedings of the FSE/SDP workshop on Future of software engineering research*, 2010, pp. 315 – 320.

[14] B. Ji, X.-W. Luo, X.-X. Peng, Y. Zhang, Y.-L. Wu, and H.-Y. Xu, "Numerical investigation of the ventilated cavitating flow around an under-water vehicle based on a three-component cavitation model," *Journal of Hydrodynamics, Ser. B*, vol. 22, no. 6, pp. 753 – 759, 2010.

[15] M. Breuer and K. Mazaev, "Comparison of DES, RANS and LES for the separated flow around a flat plate at high incidence," *International Journal for Numerical Methods in Fluids*, vol. 41, no. 4, pp. 357 – 388, 2003.

[16] C. J. Mordhorst, "Investigation of open-source CFD software on shipyards-analysis of combustion outlets on large yachts using open-Foam," 2011.

[17] V. Hidalgo, X. Luo, R. Huang, and E. Cando, "Numerical simulation of cavitating flow over 2d hydrofoil using openfoam adapted for debian operating system with lxde based in kernel gnu/linux, submitted," *Proceedings of the ASME 2014 4th Joint US-European Fluids Engineering Division Summer Meeting and 11th International Conference on & Nanochannels, Microchannels, and Minichannels*, no. 4, 2014.

[18] R. Yakushiji, H. Yamaguchi, and K. Takafumi, "Investigation for unsteady cavitation and re-entrant jet on a foil section," *Japan Journal of the Society of Naval Architects*, vol. 2001, no. 190, pp. 61 – 74, 2001.