

Sensitivity Analysis of Zwart-Gerber-Belamri Model Parameters on the Numerical Simulation of Francis Runner Cavitation

¹Xavier Escaler*; ¹Rafel Roig; ²Víctor Hidalgo

¹Universitat Politècnica de Catalunya, Barcelona, Spain; ²Escuela Politécnica Nacional, Quito, Ecuador

Abstract

A series of numerical simulations of the 3D cavitation development in the GAMM Francis turbine runner were carried out with the Zwart-Gerber-Belamri (ZGB) two-phase model available in ANSYS® CFX v16.2. The aim of the investigation was to evaluate the influence of the ZGB parameters on the location, size and shape of the simulated cavitation. To begin, a sector of the entire fluid domain comprising a single blade was created taking profit from the rotational symmetry of the geometry and a stage simulation was set to simulate the flow field without cavitation. A grid independence analysis was carried out using the shear-stress transport (SST) turbulence model. Results such as the pressure drop and the torque at the best efficiency point (BEP) compared reasonably well with the experimental values, thus proving the model validity. The cavitation inception and development was simulated at BEP using the ZGB default parameters by progressively decreasing the sigma value. As expected, the onset of cavitation took place at the blade suction side close to the junction with the band, and the length and area of the blade cavity increased for decreasing sigma values. The cavity shape also showed a good agreement with the observations at the GAMM turbine model. Finally, the values corresponding to the mean nucleation site diameter, the nucleation site volume fraction and the empirical coefficients of evaporation and condensation were modified individually while keeping the rest constant in order to observe the changes in cavitation behavior. As a result, it was concluded that all the parameters have a significant effect in the cavity length, the pressure distribution and the torque. Moreover, extreme values leading to unrealistic results were also found.

Keywords: Francis runner, GAMM turbine, stage simulation, Zwart cavitation model, empirical constants, nucleation diameter

Introduction

Cavitation in hydraulic turbines is well known to be harmful because it can reduce the performance and provoke vibrations and erosion of the solid components. The turbine design combined with particular operating conditions such as the actual setting level of the unit and the deviation from the best efficiency point (BEP), among others, are the main factors that will determine the risk of cavitation and its consequences. Moreover, the cavitation phenomenon inside a turbine occurs under unsteady flows with 3D effects. For that, numerical and experimental investigations are still required to understand the physics of the phenomenon [1]. The computational fluid dynamics (CFD) software ANSYS® CFX v16.2 incorporates the unsymmetrical cavitation model proposed by Zwart-Gerber-Belamri (ZGB) [2], which is based on the interphase mass transfer, \dot{m} , derived from the Rayleigh-Plesset equation. The ZGB model is expressed by (1):

$$\dot{m} = \begin{cases} -F_v \frac{3r_{nuc}(1-\alpha)\rho_v}{R_B} \sqrt{\frac{2P_v-P}{3\rho_l}} & \text{if } P < P_v \\ F_c \frac{3\alpha\rho_v}{R_B} \sqrt{\frac{2P-P_v}{3\rho_l}} & \text{if } P > P_v \end{cases}, \quad (1)$$

where F_v and F_c are the empirical calibration coefficients for vaporization and condensation respectively, r_{nuc} is the nucleation site volume fraction and R_B is the typical bubble radius of the nucleation site in water [3]. More specifically, the model default parameters are $F_v = 50$, $F_c = 0.01$, $r_{nuc} = 5 \cdot 10^{-4}$ and $R_B = 2 \cdot 10^{-6}$. The rest of variables are the vapor volume fraction, α , the liquid density, ρ_l , the vapor density, ρ_v , the time averaged pressure, P , and the liquid saturation pressure, P_v . The ZGB model has already been used and validated in several investigations for the numerical prediction of sheet cavitation around hydrofoils [4, 5], centrifugal pumps [6], marine propellers and Kaplan turbines [7] with fairly good results. Nevertheless, in all the cases it has been necessary to tune the model empirical coefficients to ensure the accuracy of the results.

In order to evaluate the influence of the ZGB model parameters on the numerical prediction of cavitation on the blades of a Francis runner, the GAMM Francis model geometry and performance [8, 9] were taken as a study case. More specifically, cavitation inception and development were simulated at the best efficiency point (BEP) which is characterized by a flow rate $Q = 0,376 \text{ m}^3/\text{s}$, a head $H = 5,957 \text{ m}$, a rotating speed $\omega = 52,36 \text{ rad/s}$, and a torque $T = 388 \text{ Nm}$.

The computational domain did not consider the distributor stay and guide vanes. The runner fluid domain was bounded upstream by a cylindrical surface located in the middle of the distributor channel and extended downstream up to a circular surface in the draft tube channel as indicated in Figure 1a. To reduce computational effort, a sector comprising only one inter-blade channel was created taking profit from the rotational symmetry of the runner and the existence of 13 blades as indicated in Figure 1b. The simulations were carried out using the Rotating Frames of Reference (RFR) and the Shear-Stress Transport (SST) turbulence models. Initially, a grid independency analysis was done with no cavitation model activated. The simulated torque and head values showed a good agreement with the experimental ones at BEP. As a result, an accurate mesh and the correct boundary conditions necessary to predict the expected behavior were established.

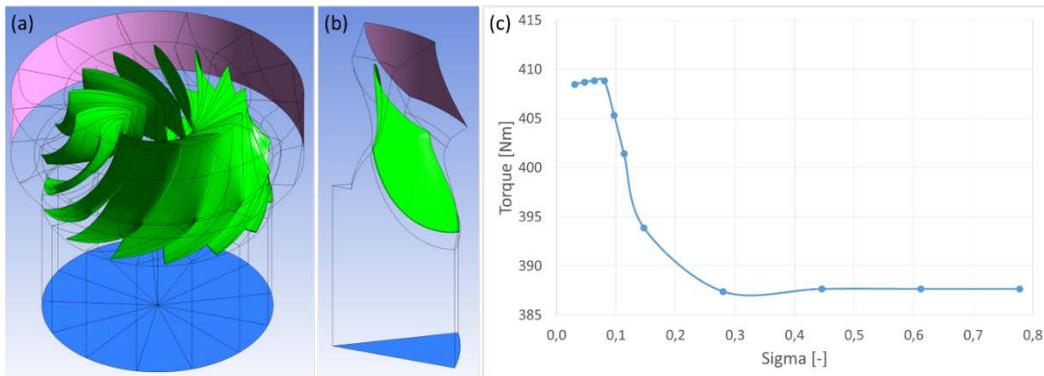


Figure 1. (a) Runner geometry; (b) sector computational domain; (c) simulated runner torque evolution.

Cavitation simulations

Initially, the cavitation development was simulated at BEP for different values of the outlet pressure, P_{out} , and using the ZGB default parameters. As a result, the typical torque rise for decreasing Sigma, σ , was correctly simulated (see Figure 1c). In our case, σ is calculated with (2) where g is the gravity:

$$\sigma = \frac{P_{out} - P_v}{\rho_l g H} \quad (2)$$

In Figure 2, the location, size and shape of the simulated leading edge cavity has been visualized with an isosurface of water vapor volume fraction equal to 0.5. As σ decreases, cavitation develops on the blades. A good similarity with the visual observations in the reduced scale GAMM turbine model at $\sigma = 0.14$ [6] is found with the numerical results at $\sigma = 0.15$, which is also in accordance with the works of Susan-Resiga *et al.* [7-8].

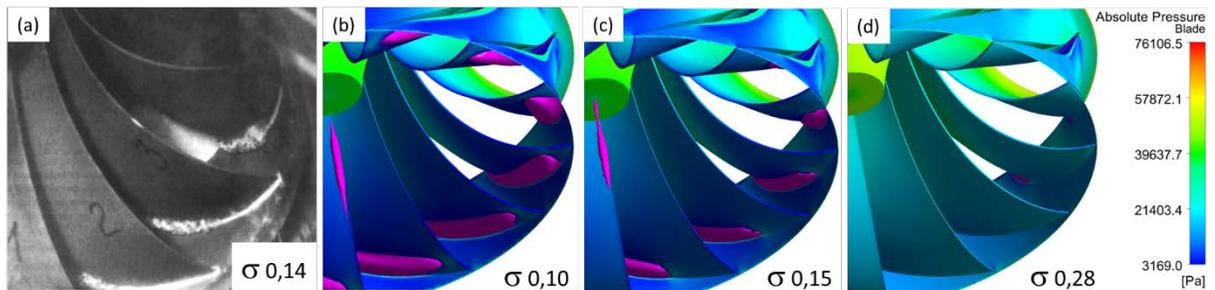


Figure 2. (a) Photography of the inlet edge cavitation in GAMM Francis turbine at BEP for $\sigma = 0.14$ [6]; contours of absolute pressure and isosurfaces of water vapor volume fraction 0.5 for $\sigma = 0.10$ (b), 0.15 (c) and 0.28 (d) using the ZGB default parameters.

	Min		Default	Max	
F_v	$5 \cdot 10^{-1}$	$5 \cdot 10^{-0}$	$5 \cdot 10^1$	$5 \cdot 10^2$	$5 \cdot 10^3$
F_c	$1 \cdot 10^{-4}$	$1 \cdot 10^{-3}$	$1 \cdot 10^{-2}$	$1 \cdot 10^{-1}$	$1 \cdot 10^0$
r_{nuc}	$5 \cdot 10^{-6}$	$5 \cdot 10^{-5}$	$5 \cdot 10^{-4}$	$5 \cdot 10^{-3}$	$5 \cdot 10^{-2}$
R_B	$2 \cdot 10^{-8}$	$2 \cdot 10^{-7}$	$2 \cdot 10^{-6}$	$2 \cdot 10^{-5}$	$2 \cdot 10^{-4}$

Table 1. Tested values of the ZGB parameters for $\sigma = 0.15$.

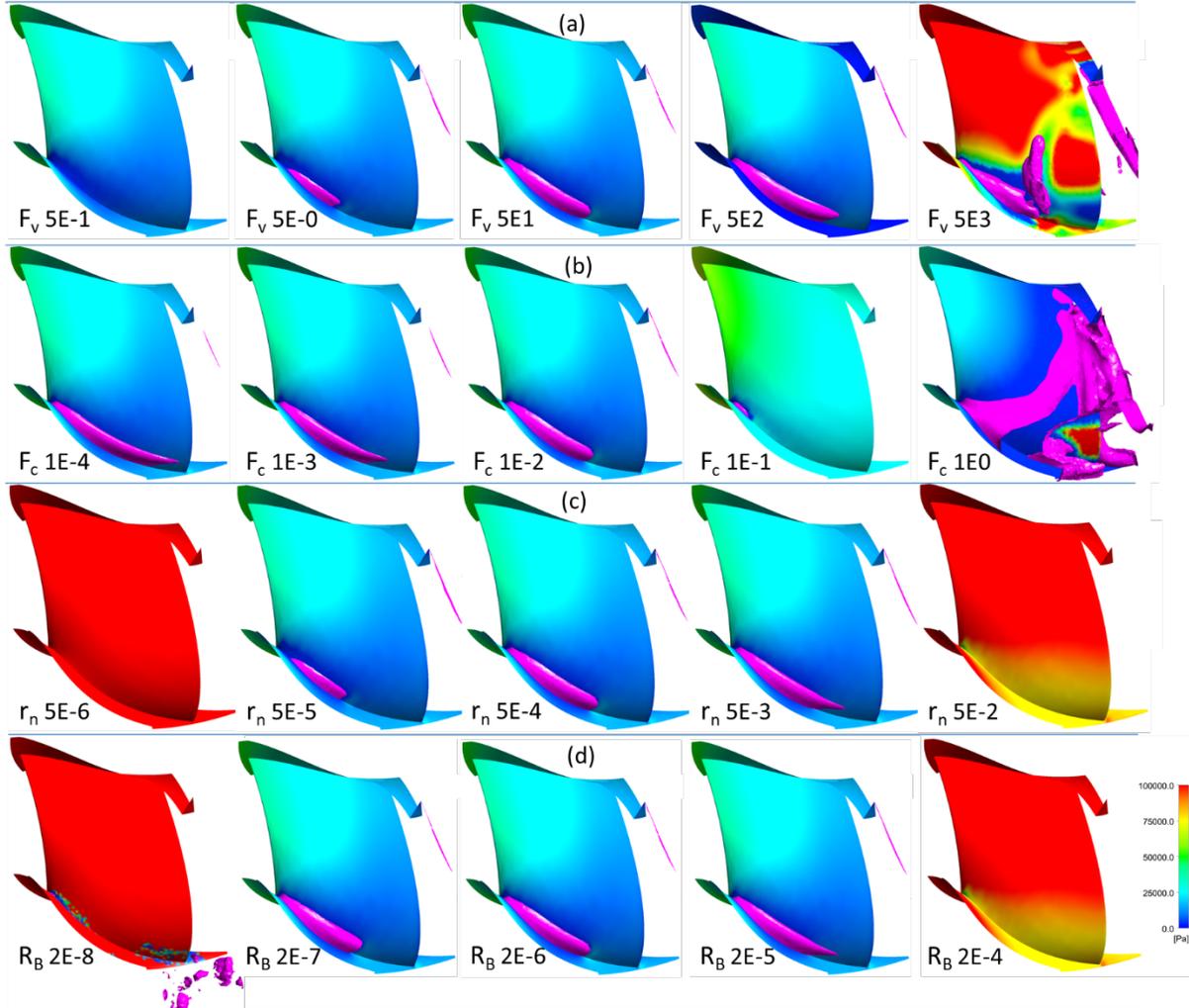


Figure 3. Contours of absolute pressure on walls and isosurface of water vapor volume fraction 0.5 at $\sigma = 0.15$ for different values of F_v (a), F_c (b), r_{nuc} (c) and R_B (d).

To study the influence of the ZGB parameters on the numerical results at $\sigma = 0.15$, each of the values indicated in Table 1 were considered individually while keeping the rest of them at their default values. The plots shown in Figure 3 correspond to the contours of absolute wall pressures and the 0.5 water vapor volume fraction isosurfaces for the different F_v (a), F_c (b), r_{nuc} (c) and R_B (d) values, from top to bottom rows respectively.

As it can be observed, unrealistic results were found for extreme values two orders of magnitude higher or lower than the default ones. In such cases, the torque can deviate more than 10% from the nominal value because the pressure distribution around the blade and the water vapor phase appear significantly altered. In particular, the possible range of values for each parameter that keep the expected runner efficiency are $[0.5 - 500]$ for F_v , $[0.0001 - 0.1]$ for F_c , $[5 \cdot 10^{-5} - 5 \cdot 10^{-3}]$ for r_{nuc} and $[2 \cdot 10^{-7} - 2 \cdot 10^{-5}]$ for R_B .

Regarding the cavitation development on the blade, all the parameters have a main influence on its length along the junction between the blade and the runner band. More specifically, F_v determines the location of both the cavitation inception and of the closure region, meanwhile F_c only determines the location of the cavitation closure. On the other hand, r_{nuc} shows a similar effect to F_v , and R_B shows a similar effect to the reciprocal of F_c .

Conclusion

The default values of the Zwart-Gerber-Belamri cavitation model available in ANSYS® CFX v16.2 appear to provide results that are in accordance with the cavitation observations in the GAMM Francis turbine runner at BEP. Nevertheless, the sensitivity of the inception and closure locations of the inlet edge cavitation to these parameters is found to be significant when they are increased or decreased by at least one order of magnitude. Moreover, unrealistic results are obtained if extreme values of the parameters are considered.

References

- [1] Ji, B., Long, Y., Long, X., Qian, Z., and Zhou, J. (2017) *Large eddy simulation of turbulent attached cavitating flow with special emphasis on large scale structures of the hydrofoil wake and turbulence-cavitation interactions*. Journal of Hydrodynamics, Ser. B, 29(1):27-39.
- [2] Zwart, P., Gerber, A. and Belamri, T. (2004) *A Two-Phase Flow Model for Predicting Cavitation Dynamics*. Proceedings of the ICMF 2004 Int. Conf. on Multiphase Flow, Yokohama, Japan.
- [3] Hidalgo, V. H., Luo, X. W., Escaler, X., Ji, J. and Aguinaga, A. (2014). *Numerical investigation of unsteady cavitation around a NACA 66 hydrofoil using OpenFOAM*. IOP Conf. Ser.: Earth Environ. Sci. 22 052013.
- [4] Hidalgo, V., Luo, X. W., Escaler, X., Bin, J. I. and Aguinaga, A. (2015). *Implicit large eddy simulation of unsteady cloud cavitation around a plane-convex hydrofoil*. Journal of Hydrodynamics, Ser. B, 27(6), 815-823.
- [5] Morgut, M., Nobile, E. and Biluš, I. (2011) *Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil*. International Journal of Multiphase Flow, 37(6):620–626.
- [6] Liu, H-l., Wang, J., Wang, Y., Zhang, H. and Huang, H. (2014). *Influence of the empirical coefficients of cavitation model on predicting cavitating flow in the centrifugal pump*. International Journal of Naval Architecture and Ocean Engineering, 6(1):119–131.
- [7] Morgut, M., Jošt, D., Škerlavaj, A., Nobile, E. and Contento, G. (2017) *Numerical Predictions of Cavitating Flow Around a Marine Propeller and Kaplan Turbine Runner with Calibrated Cavitation Models*. Strojniški vestnik – Journal of Mechanical Engineering. Article in Press.
- [8] Avellan, F., Dupont, P., Farhat, M., Gindroz, B., Henry, P., Hussain, M., Parkinson, E. and Santal, O. (1990) *Flow Survey and blade pressure measurements in a Francis turbine model*. Proceedings of 15th IAHR Symposium on modern technology in hydraulic energy production, September 1990, Belgrade, vol. 2, Session I, Paper 14, 14 pages.
- [9] Avellan F., Dupont P., Farhat M., Gindroz B., Henry P. and Hussain M. (1993) *Experimental flow study of the GAMM turbine model*. Proceedings of the GAMM Workshop “3D-computation of incompressible internal flow” Ed. Sottas G. and Ryhming I.L., NNFM 39, Vieweg Verlag, Braunschweig, pp. 33-53.
- [10] Susan-Resiga, R., Muntean, S. and Anton, I. (2002) *Numerical Analysis of Cavitation Inception in Francis Turbine*. Proceedings of the Hydraulic Machinery and Systems 21st IAHR Symposium, Lausanne, Switzerland.
- [11] Susan-Resiga, R., Muntean, S., Anton, I. and Bernard, S. (2003) *Numerical Investigation of 3D Cavitating Flow in Francis Turbines*. Conference proceedings / Conference on Modelling Fluid Flow (CMFF '03) The 12th Int. Conf. on Fluid Flow Technologies, Budapest, Hungary.