3D CAVITATING FLOW IN HYDRAULIC FRANCIS TURBINES

Sebastian MUNTEAN, Sandor BERNAD,
Romeo RESIGA, Ioan ANTON

Abstract: We present a numerical investigation of the cavitating flow in a Francis turbine runner. First, the steady non-cavitating relative flow is computed in a runner interblade channel using the mixing interface approach. Second, the cavitation model is activated. Results for cavity shape and extent, as well as for the pressure distribution on the blade with and without cavity are presented and discussed.

NOMENCLATURE

c_p	[-]	pressure coefficient
g	$[m/s^2]$	gravity
$\dot{m}$	$[kg/(s.m^3)]$	inter-phase mass flow rate per unit volume
n_b	$[1/m^3]$	number of bubbles per unit volume of liquid
p	$[Pa]$	pressure
E	$[J/kg]$	specific energy (energy per unit mass)
H_s	$[m]$	suction head
α	[-]	vapour volume fraction
ρ	$[kg/m^3]$	density
σ	[-]	cavitation number

SUBSCRIPTS AND SUPERSCRIPTS

m	mixture
v	vapour or vaporization
l	liquid
ref	reference point
atm	atmospheric conditions
PS, SS	pressure side, suction side

1. INTRODUCTION

Cavitation is one of the main problems in hydraulic turbines and pumps, since it produces strong vibrations and noise, as well as erosion. Usually, the cavitating behavior of a turbomachine is evaluated in the design or analysis stage based on a liquid flow numerical simulation. In doing so, one identifies the regions with pressure smaller than the vaporization pressure, and estimates the cavity size and location. However, this approach does not take into account the influence of the cavity presence on the flow field. As a result, conclusions obtained from liquid flow simulation can be useful to evaluate the cavitation inception, or to estimate at most the early cavitational stages.

Computing two-phase cavitating flows is a challenge since the cavitating bubbles, or bubble clouds, have a very complicated dynamics, including inter-phase phenomena. Tracing the gas-liquid interface might be possible for a cavitational bubble, or for a configuration of several bubbles, but this approach is total impractical for industrial application. Moreover, the engineer does not need to know the flow evolution at this level of details. As a result, a local averaging procedure, that considers a homogeneous liquid-vapor mixture, is a reasonable approach as far as the computing time is concerned. We have presented such a method in [6], and the comparison with experimental data for simple axisymmetric cavitators is very good. Since it seems that the approach we have employed so far is suitable for stable attached cavitation, at least, we are using this methodology to investigate the cavitating flows in hydraulic Francis turbine.

The turbulent steady relative flow is first computed for a Francis turbine runner. The runner flow field is obtained by coupling the runner relative flow with the turbine distributor absolute flow through a mixing interface [5]. The mixing algoritm [4] averages the velocity and pressure fields in circumferential direction. As a result, there is no unsteady interaction between the guide vane wakes and runner blades, and the runner flow is steady.

Once a liquid steady relative flow is computed for the runner, the cavitating model is switched on. As a result, in regions where the pressure is smaller than the vaporization value a liquid-vapor mass flux is introduced, and the liquid is gradually turned into vapor. As a result, a steady cavity is formed and the liquid flow is locally altered. Moreover, the pressure field is significantly changed since the minimum pressure is increased (toward the vaporization pressure). The pressure field in the liquid phase is modified in the cavity neighborhood since the liquid practically flows around a body with a modified shape that includes the cavity.

Section 2 of this paper presents a set of numerical results for the cavitating flow in the GAMM Francis turbine runner operated at the best efficiency point. Section 3 summarizes the main conclusions of the present study.

2. CAVITATING FLOW IN GAMM FRANCIS TURBINE RUNNER

The operating conditions are set to achieve a cavitation number

$$\sigma = \frac{p_{atm} - p_v - \rho_l g H_s}{\rho_l E} = 0.1$$

with $p_{atm} = 96200\,Pa$, $p_v = 1285\,Pa$, $H_s = 9.06\,m$ and $E = 60.33\,J/kg$. Note that the suction head H_s is adjusted in the test rig by adjusting the tank pressure.

The pressure coefficient is defined for GAMM Francis turbine as

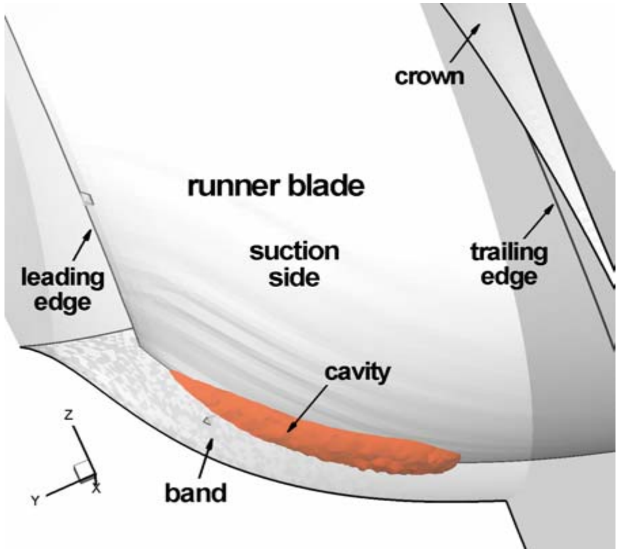


Figure 1. Cavity shape at $\sigma = 0.1$, presented as an iso-surface of $\alpha = 0.5$.

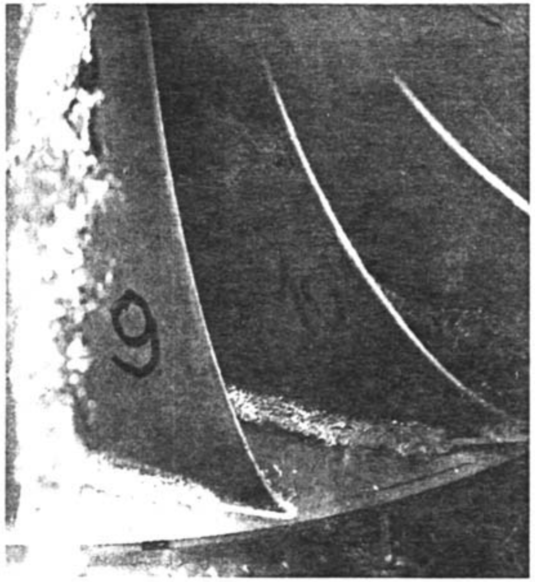


Figure 2. Cavitating flow visualisation for the GAMM Francis turbine, at $\sigma = 0.14$, [2].

$$c_p = \frac{p - p_{ref}}{\rho_l E_{ref}},$$

where p_{ref} is the pressure measured at the wall in a reference section downstream the runner, corresponding to the draft tube inlet [5].

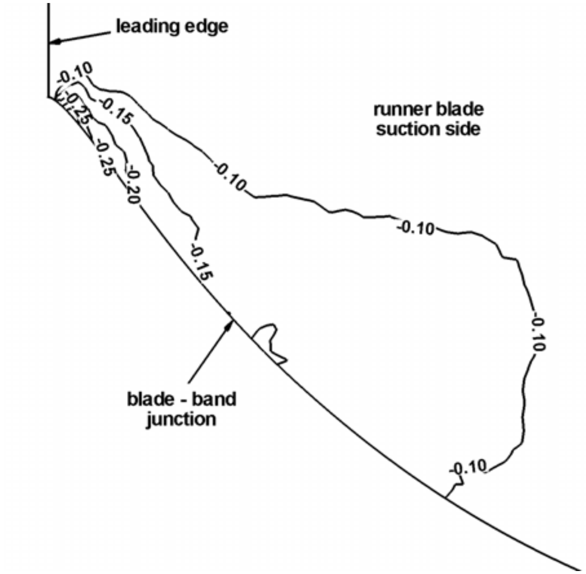
In order to evaluate the 3D shape and extend of the cavity, we are presenting in Figure 1 the iso-surface of $\alpha = 0.5$. Of course, this is only a qualitative assessment of the cavity boundary, as one may choose another iso-surface as the cavity boundary. Nevertheless, the position, shape and size of the cavity seems to be in good agreement with the cavitating flow visualisation in GAMM Francis turbine runner, Figure 2.

Note that the flow visualisation shows a travelling-cloud cavitation, where distinct bubbles can still be observed, Although the mixture model used here does not account for individual bubbles, the fact that α does not exceed 0.6 inside the cavity shows that there are no parts of the cavity completely filled with vapour.
The main advantage of directly computing the cavitating flows is that one can evaluate the influence of the cavity on the pressure distribution on the blade and further on the runner torque and turbine efficiency. Figure 3a shows the pressure distribution on the runner blade suction side, near the leading edge and at the blade-band junction, where no cavitation is presented (liquid flow). Several lines of constant c_p are shown. The cavitation inception would occur theoretically if c_p drops below.

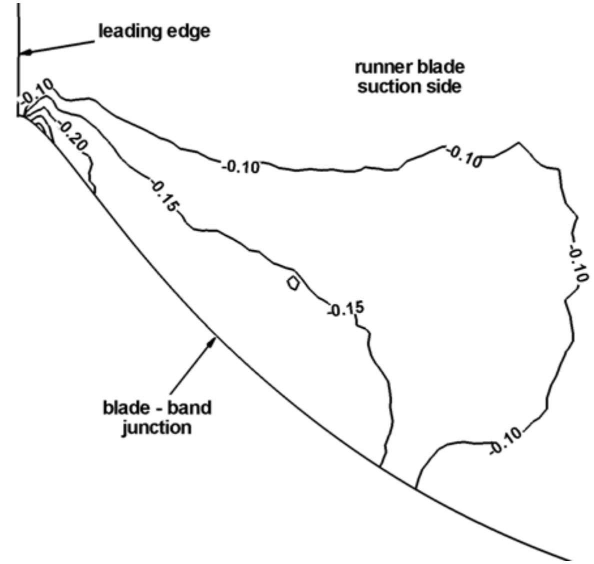
$$c_p = \frac{p_v - p_{ref}}{\rho_l E_{ref}} = \frac{1285 - 7013}{1000 \cdot 58.43} = -0.098$$

Note that the above reference pressure $p_{ref} = 7013\,Pa$ corresponds to a gauge reference pressure of $p_{ref}\big|_{gauge} = -89187\,Pa$.

The pressure of cavitation significantly changes locally the pressure distribution on the blade suction side. One can see from Figure 3b that the lines of $c_p = -0.1$ and $c_p = -0.15$ significantly differ from those in Fig. 3a. The small distance between the two lines at intersection with blade-band junction in Fig. 3b corresponds to the relatively abrupt end of the cavity.



a) flow without cavitation



b) cavitating flow with $\sigma = 0.1$

Figure 3. Detail of the pressure coefficient distribution on the runner blade suction side.

Measurements are available for c_p on the blade,on three selected sections. The section closest to the band is S15 [2]. Although S15 barely reaches the cavity, one can see from Figure 4 that there are some small changes on the pressure induced by cavitation.

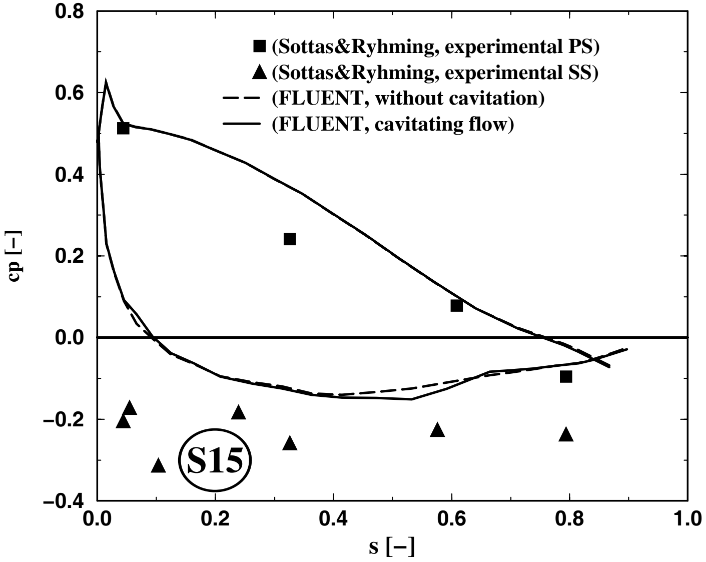


Figure 4. Pressure coefficient distribution on the suction S15 of the runner blade (near the band)

The disagreement between the measured values on the suction side of S15 and numerical results has been debated in several papers so far, but it seems that no conclusion has emerged yet. For $\sigma = 0.1$ the computed runner torque slightly increases from 374.3 N.m (liquid flow) to 375.6 N.m (cavitating flow). This is in good agreement with the turbine efficiency behaviour in the initial stage of the tolerated industrial cavitation regimes.

When the cavitation coefficient σ is further decreased, the cavity size (estimated as an iso-surface of vapour volume fraction $\alpha = 0.5$) increases. Figure 5 shows the cavity at $\sigma = 0.075$, and Figure 6 at $\sigma = 0.05$. Moreover, Figure 6 cleary shows the onset of the vortex rope on the crown.

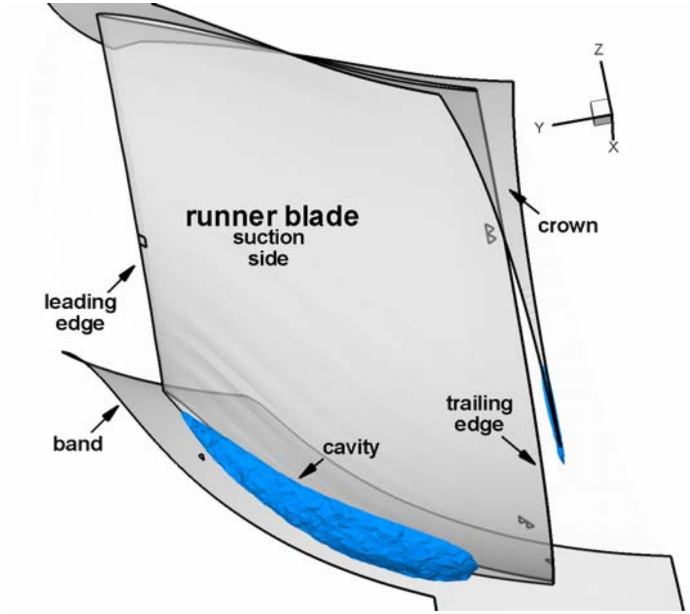


Figure 5. Cavity shape at $\sigma = 0.075$, presented as an iso-surface of $\alpha = 0.5$.

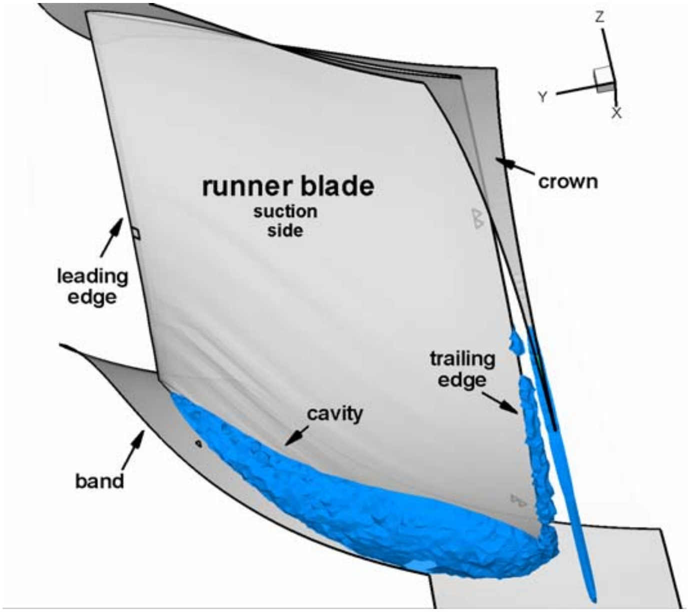


Figure 6. Cavity shape at $\sigma = 0.05$, presented as an iso-surface of $\alpha = 0.5$.

CONCLUSIONS

The paper presents a numerical investigation of the 3D cavitating flow in a Francis turbine runner, using the homogeneous mixture model. After a sigle phase (liquid) relative steady flow is computed in the runner, the cavitation model is switched on. Numerical results presented here show the ability of the cavitation model, implemented using a User-Defined-Function in the FLUENT code to correctly reproduce the cavity location and extent.

The analysis of the pressure field without and with cavitation clearly demonstrates that for advanced cavitation stages one needs to compute a two-phase flow, otherwise the pressure field cannot be correctly estimated.

The results we have obtained so far correspond to a steady flow, and therefore are valid for stable attached cavities. Further investigations will account for possible unsteadiness due to the cavity grow-detachment-collapse cycle.

REFERENCES

1. Anton, I., 1984 (Vol. 1), 1985 (Vol. 2), *Cavitation* (in Romanian), Romanian Academy Publishing House, Bucharest.
2. Avellan F., Dupont P., Farhat M., Gindroz B., Henry P., Hussain M., Parkinson E., Santal O., 1990, *Flow survey and blade pressure measurements in a Francis turbine model*. Proceedings of the XV IAHR Symposium, Belgrade, Yugoslavia, **2**, I5, 1-14.
3. Avellan F., Dupont P., Farhat M., Gindroz B., Henry P., Hussain M., 1993, *Experimental flow study of the GAMM turbine model*. In Sottas G. and Ryhming I.L., (eds.), 3D-computation of incompressible internal flows, NNFM 39, 33-53, Vieweg Verlag, Braunschweig.
4. Muntean S., Susan-Resiga S., Anton I., 2002, *3D Flow Analysis of the GAMM Francis Turbine for Variable Discharge*, in Proceedings of the XXIst IAHR Symposium on Hydraulic Machinery and Systems. Eds. Avellan F., Ciocan G. and Kvicinsky S., 9-12 September 2002, Lausanne, Switzerland, **1**, 139-146.
5. Muntean S., Resiga R., Bernad S., Anton I., 2003, *Mixing Interface Method for 3D Turbulent Flow Analysis Applied to Francis Turbine*, in I. Anton et al. (eds.) Proceedings of the Workshop on Numerical Methos in Fluid Mechanics and FLUENT Applications, May 22-23, Timişoara, 2003. (submitted)
6. Resiga R., Bernad S., Muntean S., Anton I., 2003, *Analysis and Development of Cavitating Flow Models and FLUENT Implementation*, in I. Anton et al. (eds.) Proceedings of the Workshop on Numerical Methos in Fluid Mechanics and FLUENT Applications, May 22-23, Timişoara, 2003. (submitted)
7. ***, 2001, *FLUENT 6. User's Guide*, Fluent Incorporated.

ACKNOWLEDGEMENTS

The authors acknowledge the support from the National University Research Council grants (CNCSIS A 109/2002 and At 220/2003) and Romanian Academy Grant GAR 362/2003. All numerical computations have been performed at the Numerical Simulation and Parallel Computing Laboratory of the “Politehnica” University of Timişoara, National Center for Engineering of Systems with Complex Fluids.

ADDRESSES

Sebastian MUNTEAN: Dr.eng, E-mail: seby@acad-tim.utt.ro
Sandor BERNAD: Dr.eng, E-mail: sbernad@mh.mec.utt.ro
Romanian Academy - Timişoara Branch, Center for Advanced Research in Engineering Sciences, B-dul. Mihai Viteazul No. 24-T, 300223, Timisoara, Romania

Romeo RESIGA: Associate Professor, PhD, E-mail: resiga@mh.mec.utt.ro
“Politehnica” University of Timişoara, Mechanical Engineering Faculty, Hydraulic Machinery Department, B-dul. Mihai Viteazul No. 1-T, 300222, Timisoara, Romania

Ioan ANTON: Professor: E-mail: anton@acad-tim.utt.ro
“Politehnica” University of Timişoara, Mechanical Engineering Faculty, Hydraulic Machinery Department, B-dul. Mihai Viteazul No. 1-T, 300222, Timisoara, Romania,
Member of the Romanian Academy: Romanian Academy - Timişoara Branch, Center for Advanced Research in Engineering Sciences, B-dul. Mihai Viteazul No. 24, 300223 Timisoara, Romania.