PREDICTING APPEARANCE OF CAVITATION IN PUMPS WITH A NUMERICAL APPROACH

Zgolli Ridha* & Nizar B. Souissi **
* M.C. Laboratoire Modélisation en Hydraulique et Environnement –LMHE – ENIT-Tunisie
E-mail :Ridha.Zgolli@enit.rnu.tn
** Researcher 'mater -MHE - Fluids and Process' –(LMHE-ENIT)

Date of issue of PHOENICS version used : 2005 version of PHOENICS 3.5.1

Introduction:
Cavitation behaviour prediction of hydraulic turbomachines and its associated performance drop is of high interest for the manufacturers and for the users. The experimental methods remain still the most employed and most powerful to characterize this phenomenon. For these reasons, an original three dimensional modelisation based on numerical internal flow simulations was elaborated in order to allow an accurate and rapid prediction of the cavitation sheet development in such machines and to estimate its influence on the performances.
In this paper we present the work of validation of the PHOENICS code applied in the case of a centrifugal pump, while insisting on the fact that will becomes possible to brink out the interactions rotor/stator and particularly the effect of presence of the volute on the cavitation phenomenon that can appear into the inter-blades channel of the rotating impeller. The pump with low specific speed, applied here is a centrifugal pump that was considered by other research [1] and [2].

Description of geometry:
The principal geometrical characteristics of this centrifugal pump are provided by the table1. It is composed of an impeller having 7 blades, a van less diffuser and volute denoted here collector. At nominal operating condition, the flow of water through the pump is Qn= 0.112m^3/s with total head Hn=31m for impeller rotating at N=1200 rpm.

	Int of impeller		Out let of impeller	
Rayon : R (mm)	R1	111	R2	210
Width of blade: b (mm)	b1	44	b2	28
Angle : β (degré)	β1	19°44'	β2	22°30'

Table 1 : geometrical characteristics of wheel of centrifugal pump

Modelisation of the internal flow with PHOENICS:
In the first step, the validation of this modelisation was done on a three dimensionnal with incompressible fluid assuption.
For the three dimensionnal method, modelling flow through the whole of pump turning with a constant angular spped, the corresponding computationnal domain is presented in figure1. The BFC technical with MBFGE was used .Three blocs were needed; the first bloc is used to constitute the rotating impeller (IX=181,IY=60,IZ=2), the second block is fixed and constitute the inlet part of diffuser (IX=181, IY=4, IZ=2) ,and the third bloc constitute the collector (IX=181,IY=20,IZ=2). In this case, the bondary conditions are;
- at inlet of pump (first IY of bloc1); relative velocity that obtained from given flow rate Q,
- at outlet of pump (last IY of bloc3), we introduce the pressure corresponding to the total head of the operating condition of pump.
Figure2-a and figure2-b illustrate the computationel domains of rotor (on 2 blocs) considered isolated and his collector (one bloc) respectively, that are considered to calculate the flow field with

the second method designed here 'with return on bondary conditions'. In this case we use the same operating point of pump and the boundary conditions will be introduced with an iterative procedure;
1-Calculate the flow field through the blade passage of impeller considered without collector using phoenics. Prepare results to be used as input to the collector domain.
2-Calculate the flow field and pressure field based on the input from the impeller calculation;
Then we repeat the tow precedent iterations until convergence with a criterion that consists in ensuring the stability of the flow in the whole computational domain of the pump. What is checked, when two profiles of pressure along the common border, corresponding to two successive iterations, become confused. The stabilization of the profiles speeds is then automatically checked. In this case, convergence is obtained with the fifth iteration.

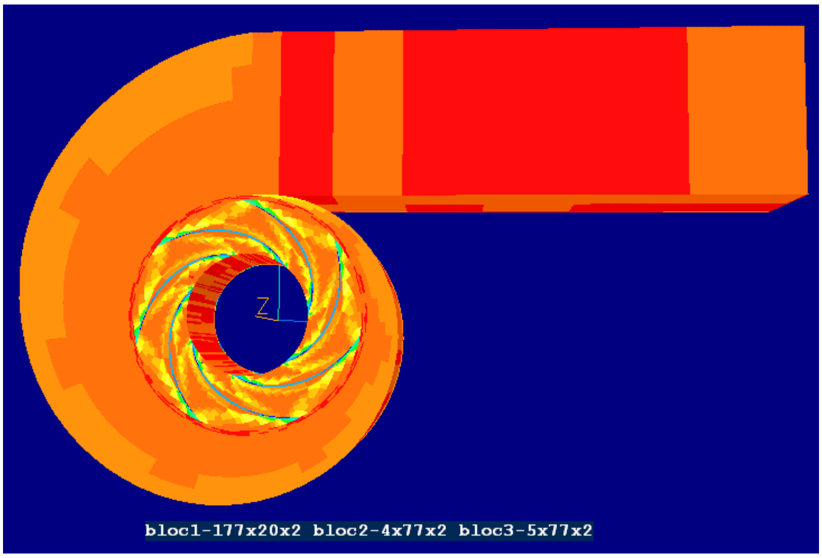

Figure1: computational domain of the pump (bloc1:177x77x2-bloc2: 77x4x2-bloc3:77x5x2)

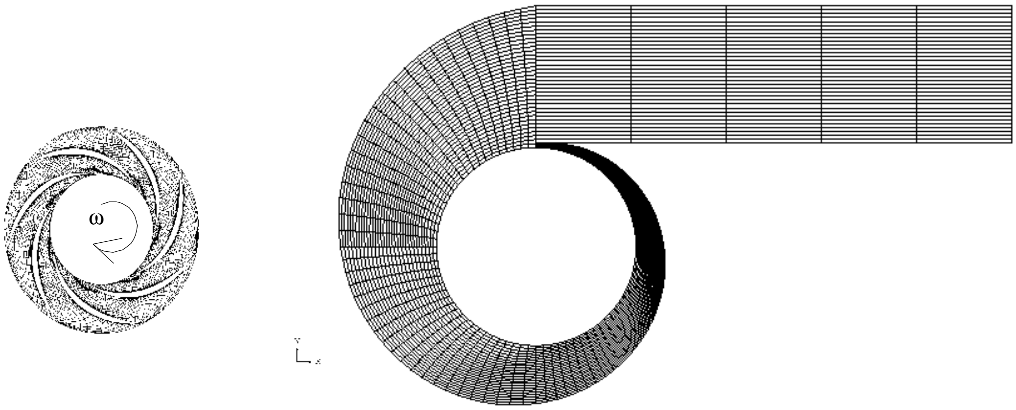

Figure 2-a) rotor Figure 2-b) collecteor;
Velocities obtained at outlet of rotor are imposed at inlet of the collector - the pressure obtained at inlet will be introduced at inlet of rotor (for the after iteration).

Results and discussions:
We begin to present the flow field calculated through the impeller considered without it's volute. For this we take off the effect induced by volute on total head of the pump.
Using the Q1-file introduced by PHOENICS, we specify in Group 2. (Transience) STEADY = F, and we restart for all time steps from result obtained with steady case.

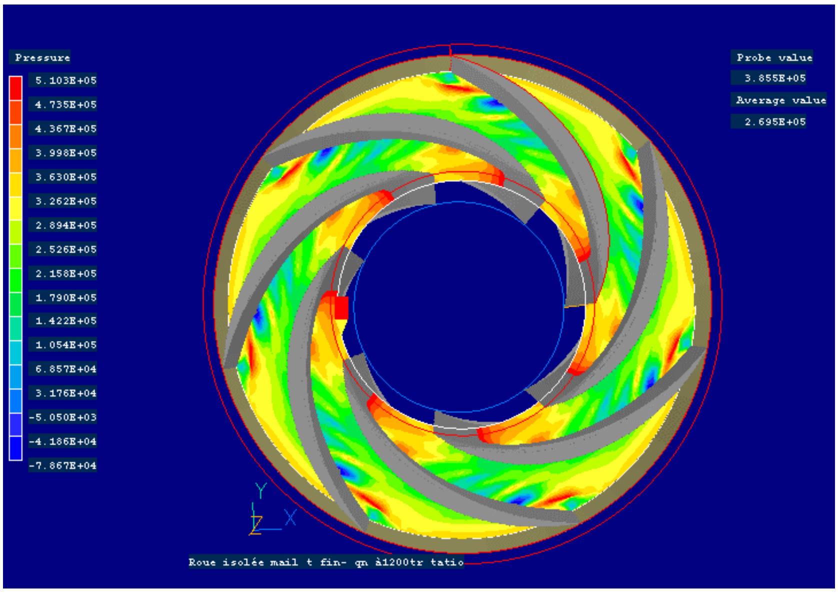

figure3 : field of pressure obtained for the flow through the impeller considered isolated

We note for the flow through the wheel considered isolated;

- A symetrical field flow , and The pressure contours clearly show an increase in pressure along the passage, as would be expected from engineering considerations of this type of flow situation
- The minimal value of pressure is located nearly the suction side of blade.

The results are in good agreement with those obtained by Azouz [1] and D. Radosavljevic [3].

Fig.4 (a-b-c-d-e-f) can illustrate that for the transient flow computed for the whole of pump starting with arbitrary initial solution, we converge to the result obtained with steady solution after 15 time steps.

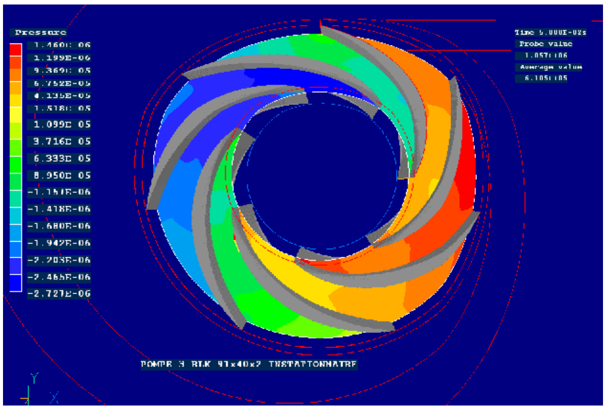

Fig.4-a: Field of pressure at time t=1s

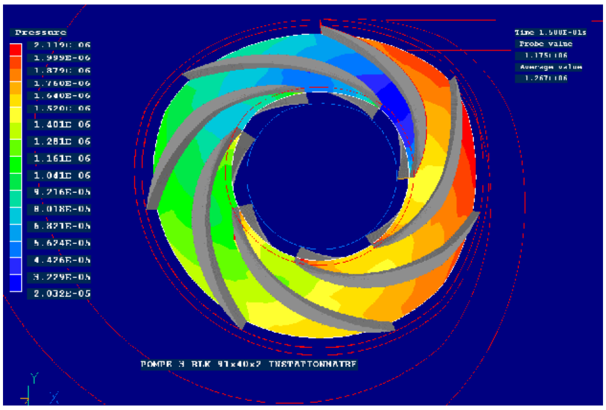

Fig.4-b: Field of pressure at time t=2s

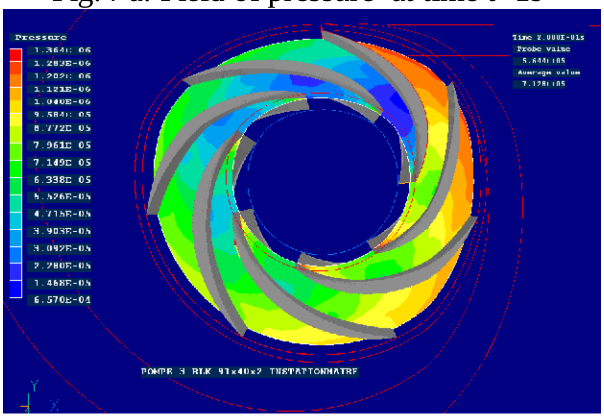

Fig.4-c Field of pressure at time t=3s

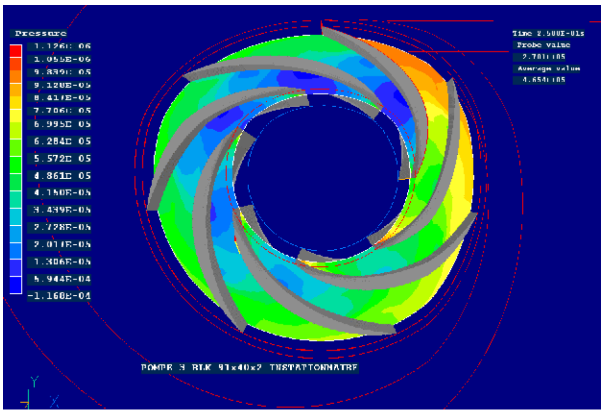

Fig.4-d Field of pressure at time t=10s

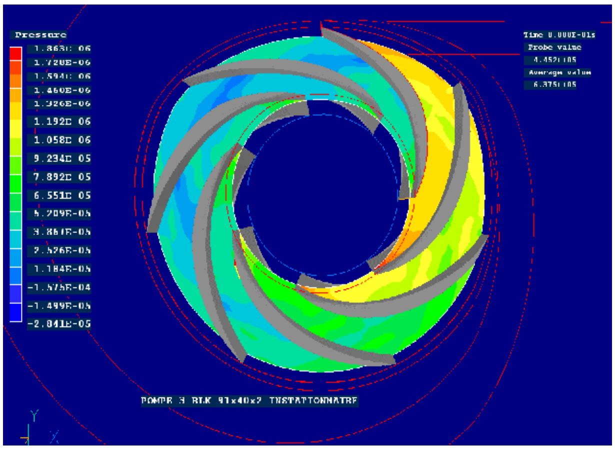

Fig.4-e: Field of pressure at time t=16s

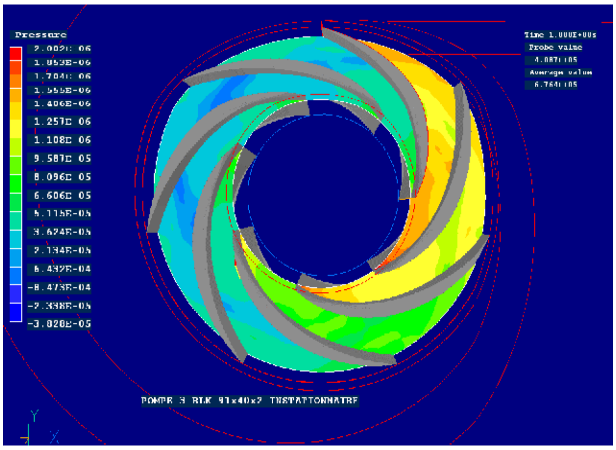

Fig.4-f: Field of pressure at time t=20s

Steady computational;

Figures 5-a and 5-b shows the respectively the pressure field and flow through the impeller obtained with consideration of it's volute at a nominal flow rate Qn. We can note the qualitative remarks as;

- field of pressure is modified by presence of volute until nearly the inlet of the rotating impeller,
- but the flow field rest unchanged
- the blade channel with minimal value of pressure, is coming before passage in front of the tongue of volute.

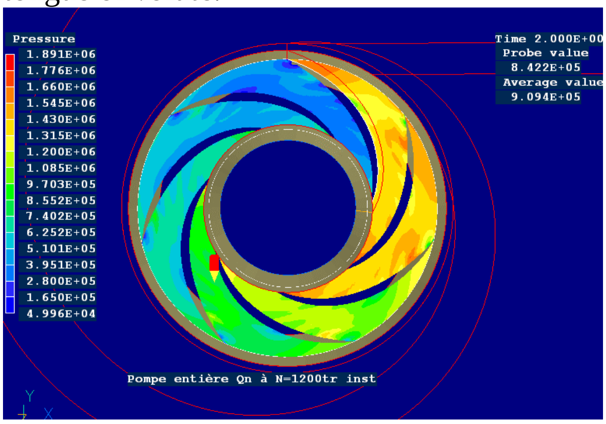

fig.5-a presure field into impeller

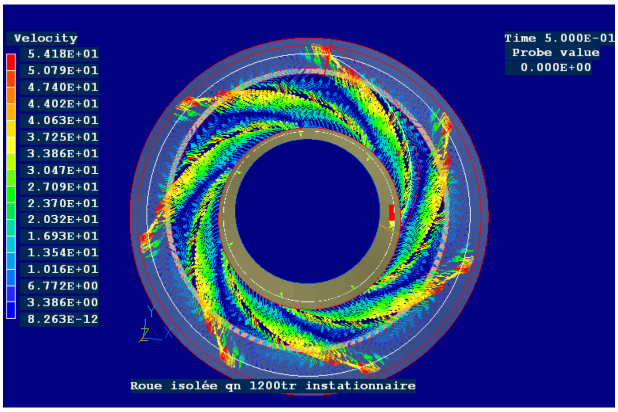

fig.5-b: relative velocity

Conclusion:

This study revealed some practical remarks, so in this way we can easily make a numerical model to approach the global performances of this hydraulic turbomachine. Here we show that the presence of volute will defer the risk of appearance of the cavitation phenomenon. And if it's produced, it will be on frequency of the rotation of wheel of the pump. We remember that the result repose on CFD with one phase flow consideration used to localize the minimum value of pressure into the channel blades of impeller.

References

[1] H.Azouz, R.Zgolli, T.Lili (2000), « Traitement numérique de la forme de la volute d'une pompe centrifuge », R.F.M.- n°1- 2000.

[2] R. Zgolli & cal. NUMERICAL APPROACH TO THE PREDICTION OF CAVITATION IN PUMPS. Fifth International Symposium on Cavitation (CAV2003) Osaka, Japan, November 1-4, 2003

[3] D. Radosavljevic ; 1999- METHOD FOR THREE-DIMENSIONAL MODELLING OF A MIXED FLOW PUMP USING PHOENICS, Phenics meeting1999.