NUMERICAL INVESTIGATION OF THE ENERGETIC AND CAVITATIONAL BEHAVIOUR OF A CENTRIFUGAL PUMP WITH DOUBLE FLUX

Adrian STUPARU, Assist. Prof.*
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

Sebastian MUNTEAN, Senior Researcher
Centre of Advanced Research in Engineering Sciences
Romanian Academy - Timisoara Branch

Alexandru BAYA, Prof.
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

Liviu ANTON, Prof.
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

*Corresponding author: Bv Mihai Viteazu 1, 300223, Timisoara, Romania
Tel.: (+40) 256 403692, Fax: (+40) 256 403700, Email: astuparu@mh.mec.upt.ro

ABSTRACT

This paper presents the numerical investigation of the 3D flow in the impeller of a centrifugal pump with double flux from a pumping station by using commercial code FLUENT 6.3. First the investigated centrifugal pump is described, then the equations that govern the flow and the boundary conditions imposed and in the end the results of the flow simulation are compared with the experimental results and a couple of conclusions are presented.

KEYWORDS

storage pump with double flux, numerical investigation, turbulent flow, energetic and cavitational behaviour analyse

NOMENCLATURE

t	[s]	time
p	[Pa]	pressure
V	[m/s]	velocity
W	[m/s]	relative velocity
ω	[rad/s]	angular speed
Q	[m^3/s]	flow rate
H	[m]	pumping head
η	[%]	efficiency
c_p	[-]	pressure coefficient
g	[m/s^2]	gravity
ρ	[kg/m^3]	density
k	[m^2/s^2]	turbulent kinetic energy

1. INTRODUCTION

The progress in the field of Computational Fluid Dynamics (CFD) has made this technology an important tool in analysis and design of hydraulic turbo machinery. The turbo machinery flow is essentially unsteady due to the rotor-stator interaction. On the other hand, rigorously speaking, the geometrical periodicity of the rotor blade channels cannot be used since there are differences in the flow from one inter-blade channel to another. However, with carefully chosen and experimentally validated assumptions, one can develop a methodology for computing the turbo machinery flow, so that very good and engineering useful results are obtained [5].

However, computing the real flow (unsteady and turbulent) through the whole storage pump requires large computer memory and CPU time even for our days computers. As a result, a simplified simulation technique must be employed to obtain useful results for pump analysis, using currently available computing resources.

The investigated storage pump is a centrifugal pump with double flux and two impellers situated in opposition and has a suction-elbow of complex geometry, Figure 1. Each impeller has five blades.

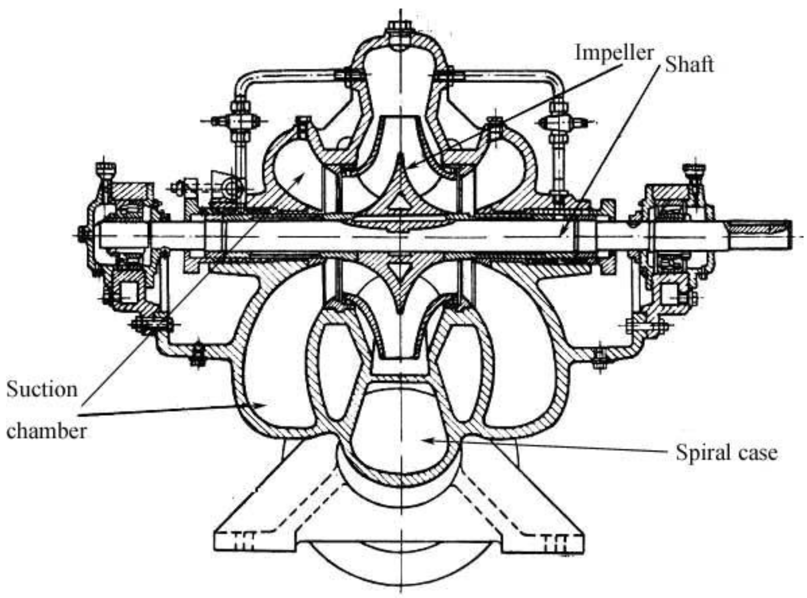

Figure 1. Storage pump with double flux

2. COMPUTATIONAL DOMAIN, EQUATIONS AND BOUNDARY CONDITIONS

The computational domain includes the suction domain and the impeller of the centrifugal pump. For the numerical investigation only one inter-blade channel is used because of the symmetry of the geometry.

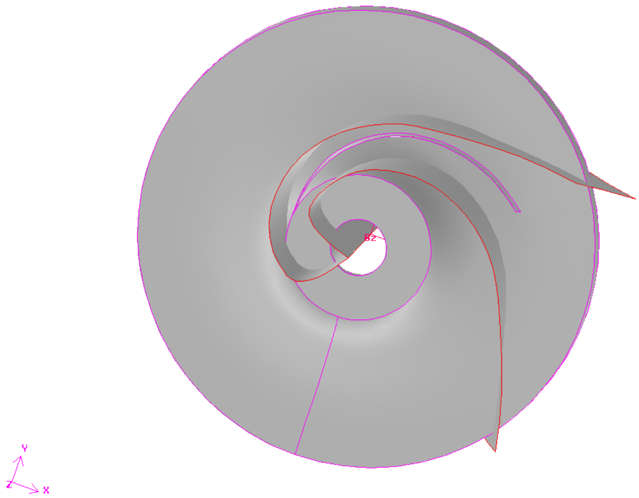

Figure 2. Isometric view of the inter-blade channel

Figure 3 shows the 3D computational domain of an inter-blade channel of the impeller. The computational domain is bounded upstream by an annular section (wrapped on the same annular surface as the suction outlet section, but different in angular extension) and extends downstream up to cylindrical patch, in order to impose the boundary conditions on the outlet section.

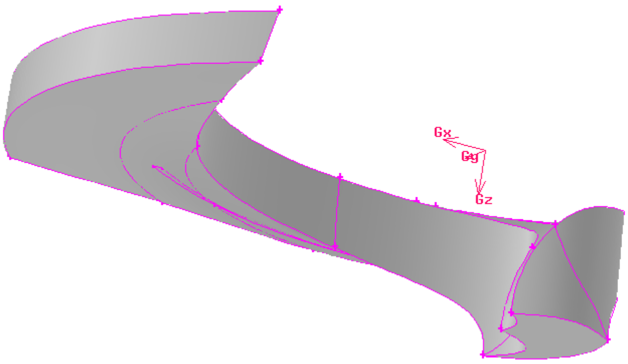

Figure 3. Computational domain of the impeller

The inter-blade channel domain is discretized with 600k cells using a structured mesh, see Figure 4.

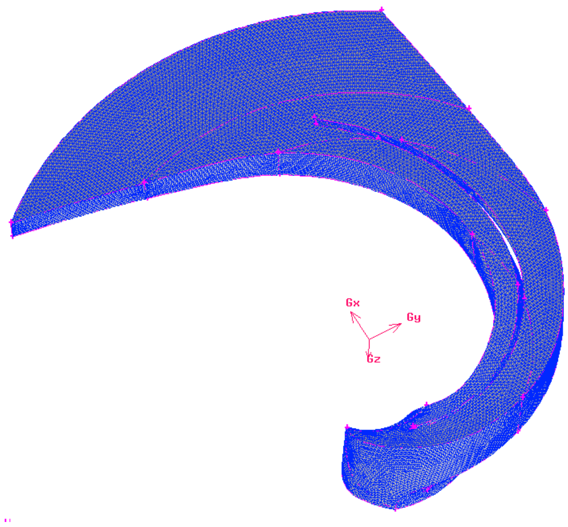

Figure 4. Mesh generated on the 3D computational domain of the inter-blade channel

Figure 5 presents the 3D computational domain of the suction pump which is bounded upstream and downstream by circular and annular section, respectively. The first one corresponds to the cross section in the suction pipe, while the second one is conventionally considered to be the suction-impeller interface.

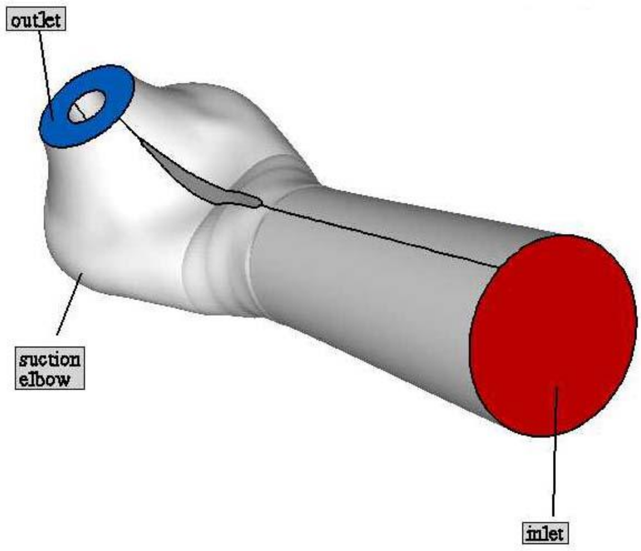

Figure 5. Isometric view of the suction domain

On the three dimensional computational domain of the suction a mixed mesh (structured and unstructured mesh) with 1.3M cells is generated, Figure 6.

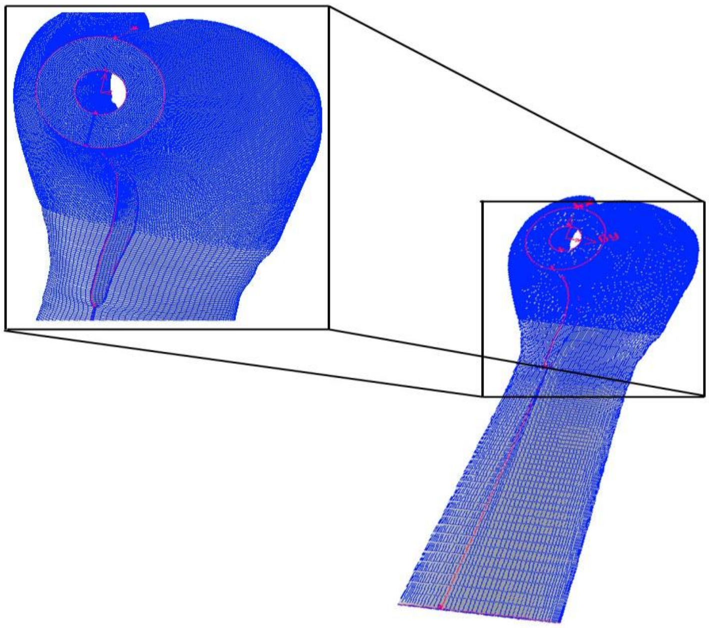

Figure 6. Mesh generated on the 3D computational domain of the pump suction.

For the flow analysis presented in this paper we consider a 3D turbulent flow model. A steady relative 3D flow is computed in the suction domain,

$$\nabla \cdot \vec{V} = 0 \tag{1}$$

$$\frac{d \;\; \rho\vec{V}}{dt} = \rho g - \nabla p + \mu\Delta\vec{V} \tag{2}$$

The numerical solution of flow equations (1) and (2) is obtained with the expert code FLUENT 6.3, [3], using a Reynolds-averaged Navier-Stokes (RANS) solver. As a result, the viscosity coefficient is written as a sum of molecular viscosity μ and turbulent viscosity μ_T, and the last term in the right-hand-side of (2) became $\nabla \cdot \left[\;\mu + \mu_T \;\; \nabla\vec{V}\right]$. For steady, absolute flow the left hand side of (2) reduces $\nabla \cdot \;\; \rho\vec{V}\vec{V}$

In the impeller domain we solve the flow equation for a relative flow, in a rotating frame of reference with angular speed $\omega = \omega\vec{k}$, $\vec{k}$ being the z axis unit vector.

By introducing the relative velocity

$$\vec{W} = \vec{V} - \vec{\omega} \times \vec{r} \tag{3}$$

with $\vec{r}$ the position vector, the left hand side of (2) became

$$
\begin{aligned}
&\frac{\partial}{\partial t}\left(\rho\vec{W}\right)+\nabla\cdot\left(\rho\vec{W}\vec{W}\right)+2\rho\vec{\omega}\times \\
&+\rho\vec{\omega}\times\left(\vec{\omega}\times\vec{r}\right)+\rho\frac{\partial\vec{\omega}}{\partial t}\times\vec{r} \qquad (4)
\end{aligned}
$$

An important assumption used in the present computation is that the relative flow is steady. This simplifies (4) by removing the first and last terms, and also allows the computation of impeller flow on a single inter-blade channel. When coupling the hydrodynamic fields of the suction domain and the impeller all circumferential variations must be removed in order to keep the relative flow steady.

The turbulent viscosity is computed using the RNG *k-ε* model.

We imposed on the inlet section of the suction domain a normal velocity, corresponding to the prescribed discharge, together with the turbulence parameters.

$$
\mathrm{v}_{\mathrm{IN}}=\frac{\mathrm{Q}}{\mathrm{S}_{\mathrm{IN}}} \qquad (5)
$$

On the outlet section of the suction domain and of the impeller domain a radial equilibrium condition is chosen.

We introduce a mixing interface technique for coupling the suction and impeller velocity, pressure and turbulence fields. This approach requires a circumferential averaging of the quantities on the suction outlet and impeller inlet sections, respectively. An iterative process is employed to obtain a continuous flow field across the suction-impeller interface. We start by computing the suction flow, with an arbitrary suction outflow pressure. The resulting velocity on the suction outlet section is circumferentially averaged, corrected in order to preserve the prescribed discharge, and plugged in as an inflow condition for the impeller. After computing the impeller flow, a new suction outflow pressure is obtained, and so on. The iterating process is stopped when no significant changes in pressure field occur from the previous iteration.

On the periodic surfaces of the impeller the periodicity of the velocity, pressure and turbulence parameters were imposed:

$$
\begin{aligned}
\vec{V}\left(r,\theta,z\right)&=\vec{V}\left(r,\theta+\frac{2\pi}{n_b},z\right)\\
p\left(r,\theta,z\right)&=p\left(r,\theta+\frac{2\pi}{n_b},z\right) \qquad (6)
\end{aligned}
$$

The remaining boundary conditions for the suction domain and the impeller domain correspond to zero relative velocity on the blade, crown and hub.

We investigated five operating points of the pump with the characteristics given in the table 1:

Table 1: Operating points of the centrifugal pump

Operating point	**Parameter**	**Symbol**	**Value**
1	Rotational speed	n [rot/min]	1000
	Flow rate	Q [m^3/s]	1.93
	Pumping head	H [m]	185
2	Rotational speed	n [rot/min]	1000
	Flow rate	Q [m^3/s]	1.97
	Pumping head	H [m]	184
3	Rotational speed	n [rot/min]	1000
	Flow rate	Q [m^3/s]	2.01
	Pumping head	H [m]	183
4	Rotational speed	n [rot/min]	1000
	Flow rate	Q [m^3/s]	2.03
	Pumping head	H [m]	183.2
5	Rotational speed	n [rot/min]	1000
	Flow rate	Q [m^3/s]	2.116
	Pumping head	H [m]	181.1

3. NUMERICAL RESULTS

The results of the numerical investigation of the flow in one impeller of the storage pump with double flux are presented in table 2.

Table 2: Numerical results for one impeller of the storage pump

Q [m^3/s]	**H [m]**	**P_h [kW]**	**P_M [kW]**	**η_h [%]**
1.93	253.187	4792.038	5127.001	93.466
1.97	250.939	4847.912	5182.879	93.537
2.01	248.734	4902.898	5237.487	93.611
2.03	247.595	4929.010	5261.973	93.672
2.116	243.031	5043.110	5376.085	93.806

The hydraulic efficiency values resulted from the numerical simulation of the flow through one impeller are compared with the efficiency values resulted from the experimental measurements and are presented in table 3 and figure 7:

Table 3: Efficiency values resulted from the numerical simulation and from experimental measurements

Operating point	Q [m^3/s]	$\eta_{measured}$ [%]	$\eta_{hFLUENT}$ [%]
1	1.93	83.9483	93.466
2	1.97	85.0087	93.537
3	2.01	84.6403	93.611
4	2.03	84.4073	93.672
5	2.116	86.5357	93.806

The difference between the measured values of the efficiency and the ones resulted from the numerical investigation of the flow can be explained by the fact that the numerical investigation took into account only one impeller and we calculated the hydraulic efficiency of this one, without considering the hydraulic losses that appears on the suction domain and on the spiral case. The values of the efficiency measured refer to the entire efficiency of the whole storage pump.

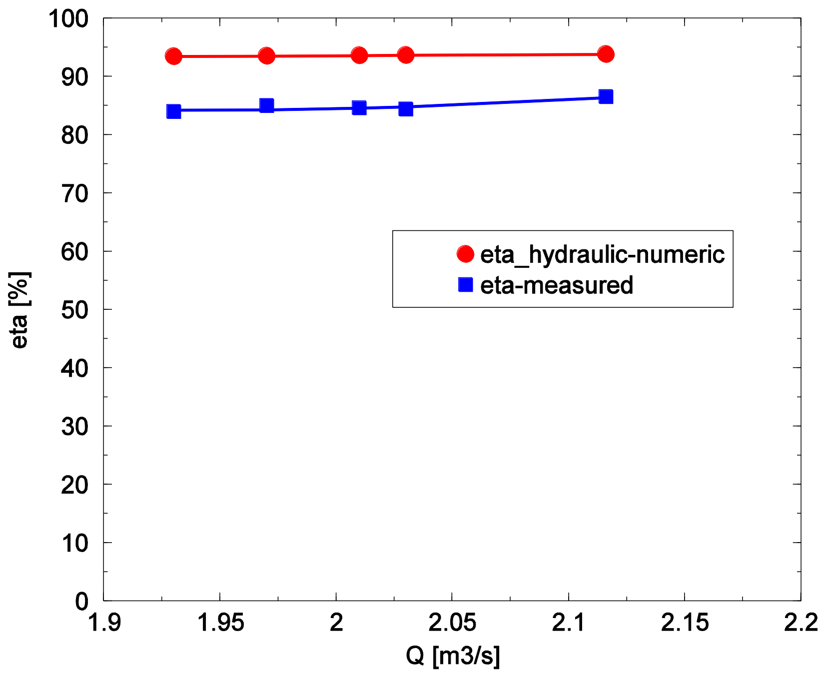

Figure 7. Efficiency vs. flow rate for the investigated operating points.

The pressure coefficient is calculated with the following relation:

$$c_p = \frac{p - p_{IN}}{\rho g H} \tag{7}$$

The distribution of the pressure coefficient along the pressure side and suction side of the blade for all five operating points are presented in figure 8 to 12.

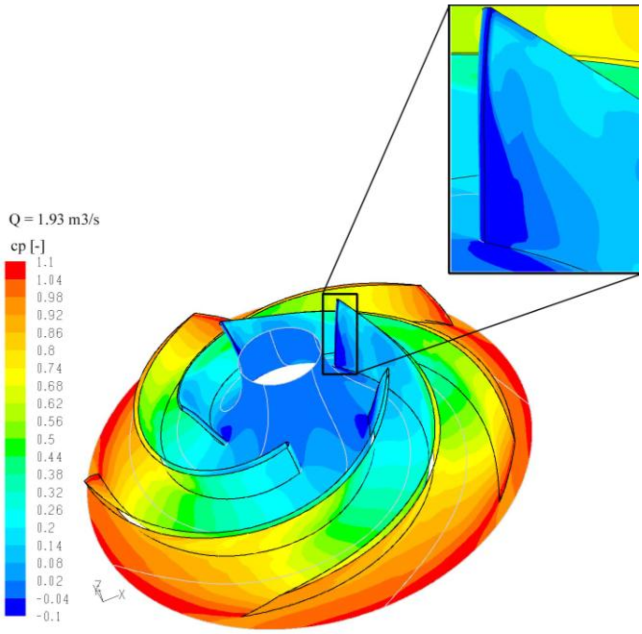

Figure 8. Pressure coefficient distribution for Q = 1.93 m^3/s

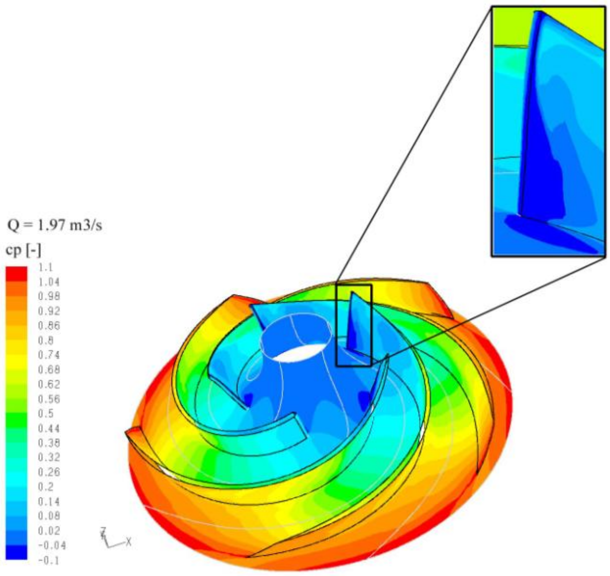

Figure 9. Pressure coefficient distribution for Q = 1.97 m^3/s

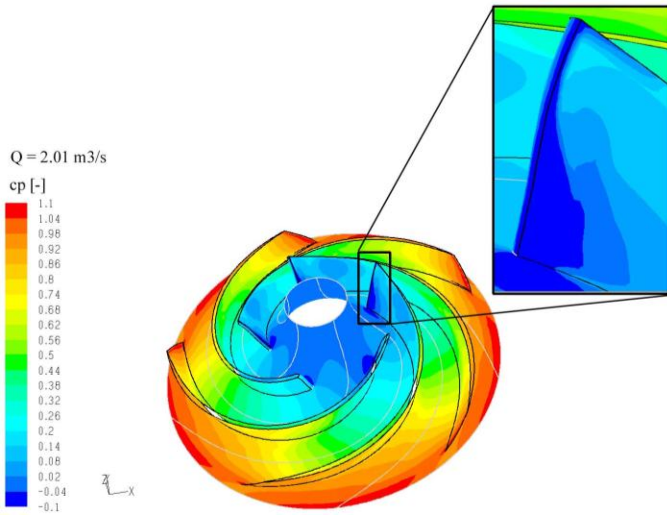

Figure 10. Pressure coefficient distribution for Q = 2.01 m^3/s

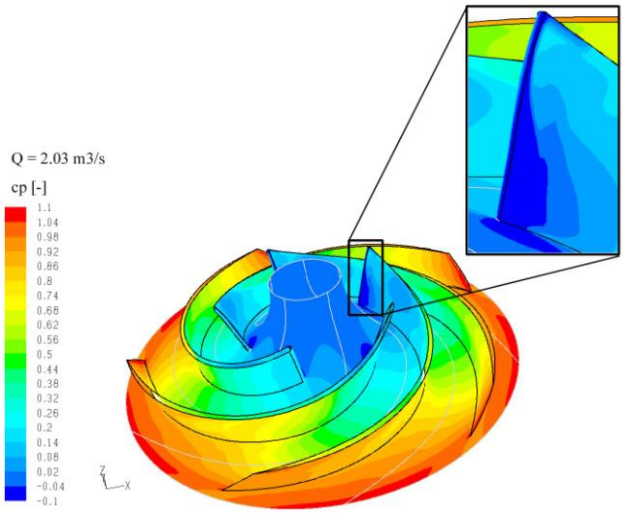

Figure 11. Pressure coefficient distribution for Q = 2.03 m^3/s

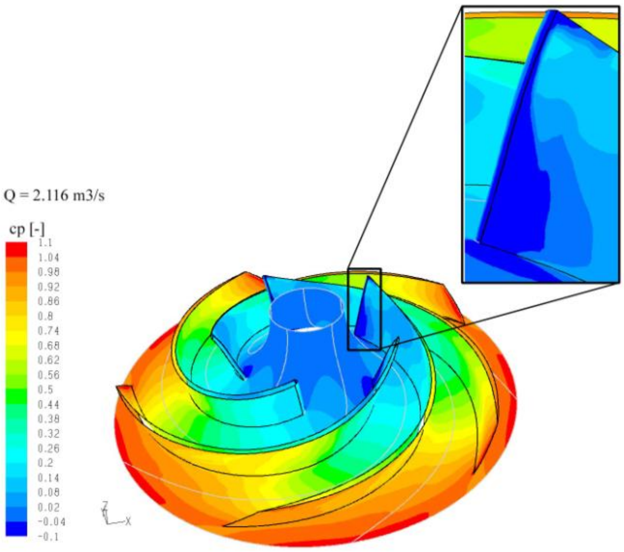

Figure 12. Pressure coefficient distribution for Q = 2.116 m^3/s

Because the investigated operating points are very close to each other, it resulted almost similar distributions of the velocity and pressure fields inside the impeller of the storage pump.

From the figures presented above one may observe that the minimum value of the pressure coefficient is reached on the suction side of the blade, near the leading edge. The blade zone with a minimum value of the pressure coefficient is larger near the hub and it is extended even on the hub itself. The minimum value of the pressure coefficient indicates the possibility of the apparition of the cavitation phenomena. This phenomena leads to the cavitational erosion of the impeller blade an d finally to the tearing of some parts of the impeller blade. The appearance of these zones with low coefficient pressure is due to the no uniformity of the flow at the inlet of the pump impeller. On the rest of the blade it is observing a good hydrodynamic load.

Analyzing the figures 8-13 it can be observe that the zones with the minimum pressure coefficient resulted from the numerical investigation of the flow are corresponding with the ones identified by the practical observation upon the impeller of the storage pump.

Figure 13. Cavitational erosion on the impeller blades of the storage pump

4. CONCLUSIONS

The paper presents a numerical study of the 3D flow in the impeller of a storage pump with double flux. Comparison between numerical results and measured data is presented.

A particular attention is paid to the analysis of the flow on the impeller blade. The numerical simulation predicts a minimum pressure on the suction side near the leading edge and towards the hub, in very good agreement with practical observations.

ACKNOWLEDGMENTS

The authors acknowledge the support from the National University Research Council grants. All numerical computations have been performed at the Numerical Simulation and Parallel Computing Laboratory of the “Politehnica” University of Timisoara, National Centre for Engineering of Systems with Complex Fluids..

REFERENCES

1. Akhras, A., El Hajem, M., Morel, R., Champagne, J.Y., *The Internal Flow Investigation of a Centrifugal Pump*, Proceeding of Flucome, Sherbrooke, Canada, 2000
2. Fluent Inc., *FLUENT 6.3 User’s Guide*, Fluent Incorporated, 2001.
3. Fluent Inc., *Gambit 2.2.30 User’s Guide*, Fluent Incorporated, 2001.
4. Hofmann, M., Stoffel, B., *Experimental And Numerical Studies On A Centrifugal Pump With 2d-Curved Blades In Cavitating Condition*, Proceeding of Fourth International Symposium on Cavitation CAV2001, Pasadena, California, SUA

5. Muntean, S., *Numerical methods for the analysis of the 3D flow in Francis turbine runners*, PhD Thesis Politehnica University of Timisoara, Timisoara, 2002.
6. Sallaberger, M., Sebestyen, A., Mannschreck, E., Pinkas, W., *Modern Pump Impeller Design With Consideration Of Non-Uniform Inlet Flow Fields*, Proceedings of IAHR, Graz, 1999
7. Tamm, A., Ludwig, G., Stoffel, B., Numerical, *Experimental And Theoretical Analysis Of The Individual Efficiencies of a Centrifugal Pump*, Proceedings of ASME FEDSM´01 2001 ASME Fluids Engineering Division Summer Meeting New Orleans, Louisiana, May 29 – June 1, 2001
8. Tamm, A., Bråten, A., Stoffel, B., Ludwig, G., *Analysis Of A Standard Pump In Reverse Operation Using CFD*, Hydraulic Machinery and Systems, 20th IAHR Symposyum, 2001
9. Van Esch, B. P. M., *Simulation of three-dimensional unsteady flow in hydraulic pumps*, Thesis University of Twente, Enschede, 1997
10. Stuparu, A., Muntean, S., Balint, D., Baya, A., Anton, L. E., *Numerical investigation of the influence of the suction-elbow on the inlet flow field of a storage pump*, Scientific Bulletin of the „Politehnica” University Of Timisoara, Transactions on Mechanics, Tom 51(65),Fascicola 3, 2006, ISSN 1224-6077, pp. 31…36
11. Stuparu, A., Muntean, S., Balint, D., Baya, A., Anton, L. E., *Numerical Analysis of Pump Hydrodynamics at Constant Speed*, Scientific Bulletin of the „Politehnica” University Of Timisoara, Transactions on Mechanics, Tom 52(66),Fascicola 3, 2007, ISSN 1224-6077, pp. 43…48