Scientific Bulletin of the
Politehnica University of Timisoara
Transactions on Mechanics
Special issue

2nd Workshop on
Vortex Dominated Flows
Bucharest, Romania
June 30 – July 1, 2006

NUMERICAL INVESTIGATION OF THE INFLUENCE OF THE SUCTION ELBOW OVER THE FLOW FIELD OF A STORAGE PUMP IMPELLER

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

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

Daniel BALINT, Assist. Prof.
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

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

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

**Corresponding author*: Bv Mihai Viteazu 24, 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 and the suction-elbow of a storage pump by using commercial code FLUENT 6.0. In pumps with suction-elbows of complex shape the interaction of the flow between the suction-elbow and the impeller is of great importance. The turn of the flow in the bend of the suction-elbow gives rise to a considerable variation of the flow at impeller inlet. Modern coupled computations of both suction-elbow and impeller take these effects into consideration. First the investigated centrifugal pump is described, then the equations that governs the flow and the boundary conditions imposed and in the end the results of the flow simulation.

KEYWORDS

storage pump, mixing interface method, turbulent flow

NOMENCLATURE

$v_r = \dfrac{V_r}{\sqrt{2gH}}$ [-] radial velocity coefficient

$v_u = \dfrac{V_u}{\sqrt{2gH}}$ [-] tangential velocity coefficient

$v_z = \dfrac{V_z}{\sqrt{2gH}}$ [-] axial velocity coefficient

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 turbomachinery. The turbomachinery 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 turbomachinery 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.

2. COMPUTATIONAL DOMAIN, EQUATIONS AND BOUNDARY CONDITIONS

The investigated storage pump has two stages with the impellers situated in opposition and a suction-elbow of complex geometry, Figure 1. Each impeller has five blades, Figure 2.

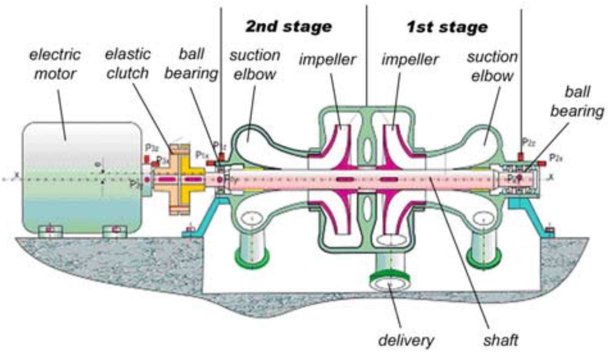

Figure 1. Cross section through the storage pump

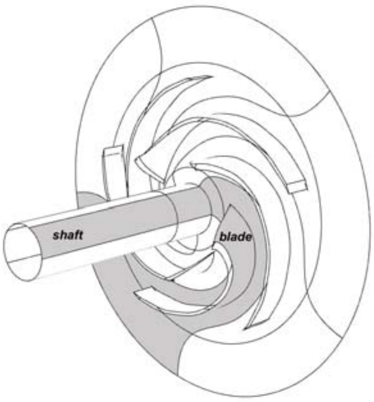

Figure 2. Isometric view of the storage pump impeller

First, the computational domain includes only the impeller of the centrifugal pump. For the numerical investigation only one inter-blade channel is used because of the symmetry of the geometry. Second, the pump computational domain is decomposed into two subdomains. The first subdomain corresponds to the suction domain includes the suction pipe as well as the suction elbow, and the second subdomain is chosen to be the impeller inter-blade channel. This approach requires a mixing interface technique to remove the circumferential variations when coupling the absolute and relative flow fields.
Figure 3 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 4 shows the 3D computational domain with boundary conditions corresponding to 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.

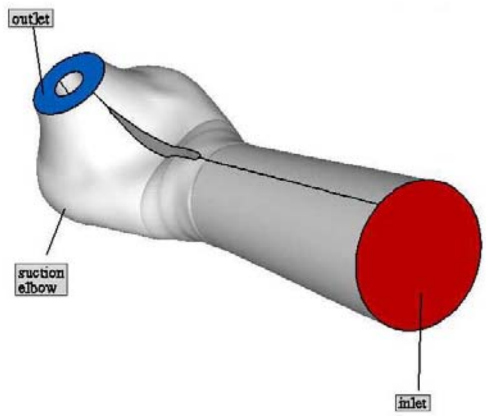

Figure 3. Isometric view of the suction domain

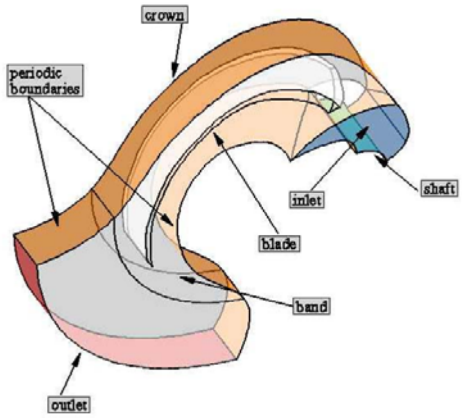

Figure 4. Isometric view of the inter-blade channel of the impeller

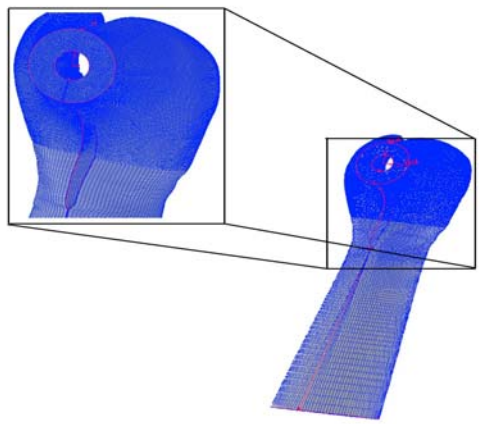

Figure 5. Mesh generated on the 3D computational domain of the pump suction

On the three dimensional computational domain of the suction a mixed mesh (structured and unstructured mesh) with 1.3M cells is generated, Figure 5. The inter-blade channel domain is discretized with 500k cells using a structured mesh coupled with a special boundary layer discretization on the blade faces, see Figure 6.

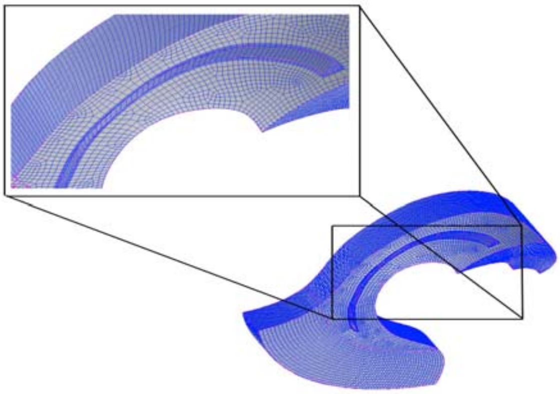

Figure 6. Mesh generated on the 3D computational domain of the inter-blade channel

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

$$\nabla \cdot V = 0 \quad (1a)$$

$$\frac{d(\rho V)}{dt} = \rho g - \nabla p + \mu \Delta V \quad (1b)$$

The numerical solution of flow equations (1a) and (1b) is obtained with the expert code FLUENT 6.0, [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 (1b) becomes $\nabla \cdot [(\mu + \mu_T)\nabla V]$.

For the first case, when only the impeller domain is considered, we solve a relative flow, in a rotating frame of reference with angular speed $\omega = \omega k$ (k being the unit vector of the pump axis direction). By introducing the relative velocity

$$W = V - \omega \times r \quad (2)$$

with r the position vector, the left hand side of (1b) becomes

$$\frac{\partial}{\partial t}(\rho W) + \nabla \cdot (\rho W W) + 2\rho\omega \times W + \\ + \rho\omega \times (\omega \times r) + \rho \frac{\partial \omega}{\partial t} \times r \quad (3)$$

An important assumption used in the present computation is that ***the relative flow is steady***. This simplifies (3) by removing the first and last terms, and also allows the computation of impeller flow on a single inter-blade channel. The turbulent viscosity is computed using the RNG model.

On the inlet surface of the impeller a constant velocity field was imposed normal on the surface. The velocity magnitude is computed using the flow rate of the operating point:

$$v_I = \frac{Q}{S_I} \quad (4)$$

For the outlet section the outflow condition is imposed. That means both flow and turbulent quantities remain unchanged downstream to the outlet section.

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

$$\vec{v}(r,\theta,z) = \vec{v}\left(r, \theta + \frac{2\pi}{n_b}, z\right), \\ p(r,\theta,z) = p\left(r, \theta + \frac{2\pi}{n_b}, z\right), \ldots \quad (5)$$

For the second case, when we take into consideration both of the domains, we imposed on the inlet section of the suction domain a normal velocity, corresponding to the prescribed discharge, together with the turbulence parameters. On the outlet section of the suction domain a radial equilibrium condition is chosen.

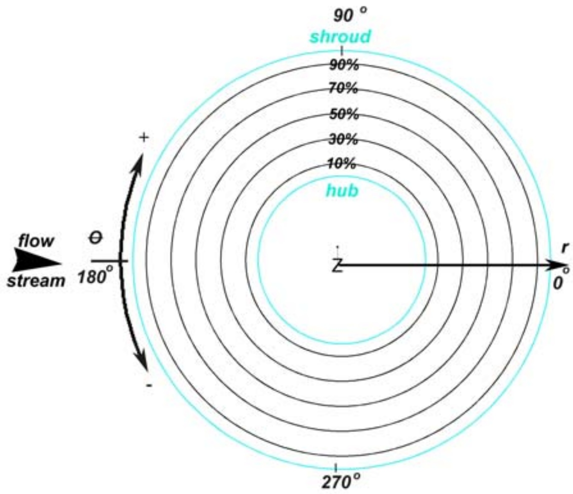

Figure 7. Coordinate system on the outlet section of the suction computational domain.

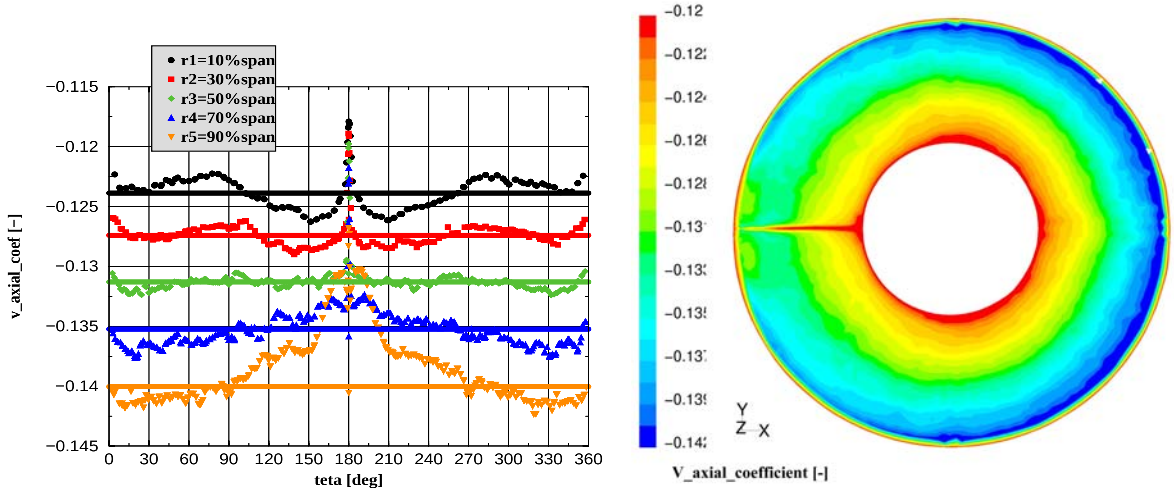

Figure 8. Axial velocity coefficient profile for the outlet section of the suction domain.

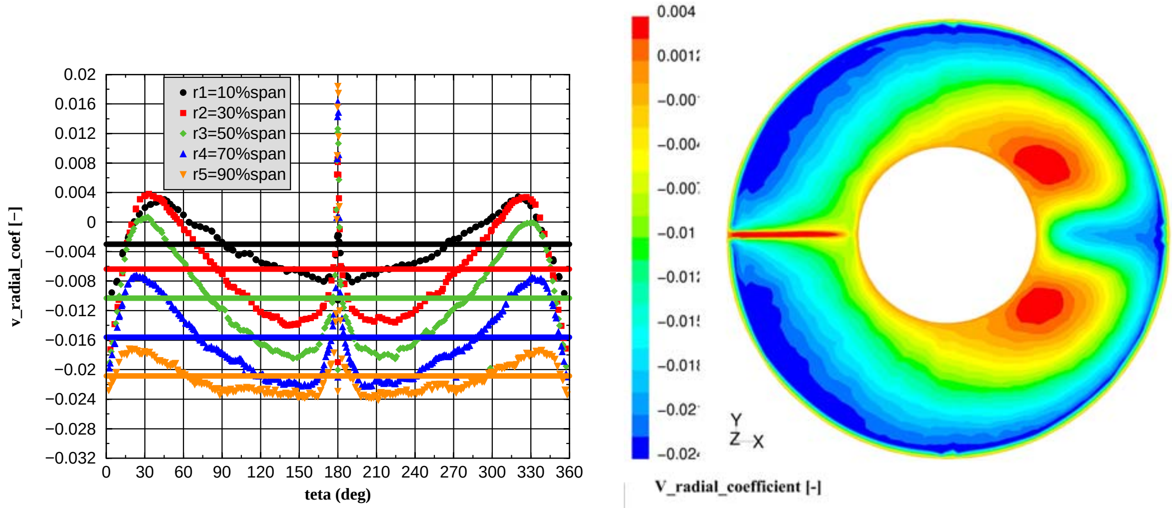

Figure 9. Radial velocity coefficient profile for the outlet section of the suction domain.

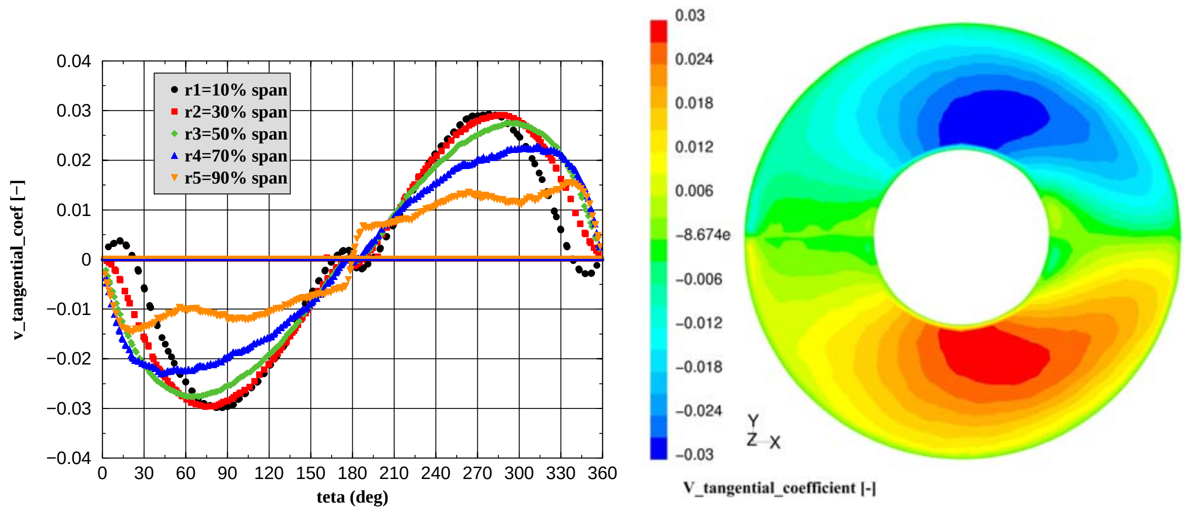

Figure 10. Tangential velocity coefficient profile for the outlet section of the suction domain.

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, as presented in figures 8 ÷ 10. 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.

3. NUMERICAL RESULTS

The $NPSH_r$ (N*et* P*ositive* S*uction* H*ead* r*equired*) is defined as:

$$NPSH_r = \left(\frac{p_i}{\rho \cdot g} + \frac{V_i^2}{2 \cdot g} \right) - \frac{p}{\rho \cdot g} \quad (6)$$

where the variables with subscript ($_i$) correspond to the averaged one on inlet section of the impeller.

In figure 11 the $NPSH_r$ distribution on the impeller blades is plotted for the case when we imposed a uniform velocity distribution on the inlet of the inter-blade channel domain:

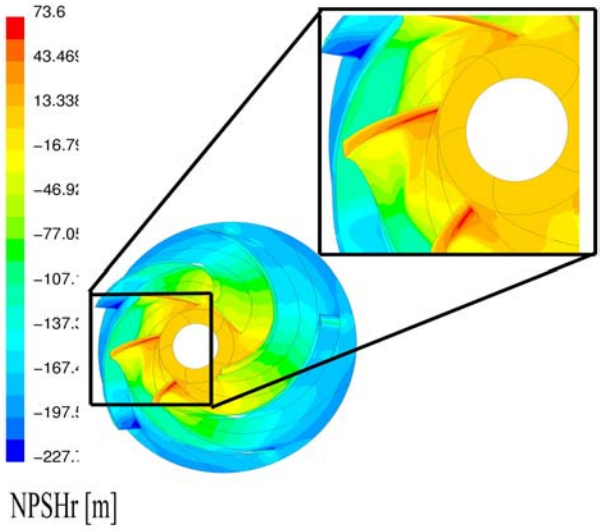

Figure 11. $NPSH_r$ distribution on the blades for uniform velocity distribution imposed on the inlet of the impeller

In figure 12 the $NPSH_r$ distribution on the impeller blades is plotted for the case when a mixed velocity field is imposed on the inlet of the inter-blade channel domain:

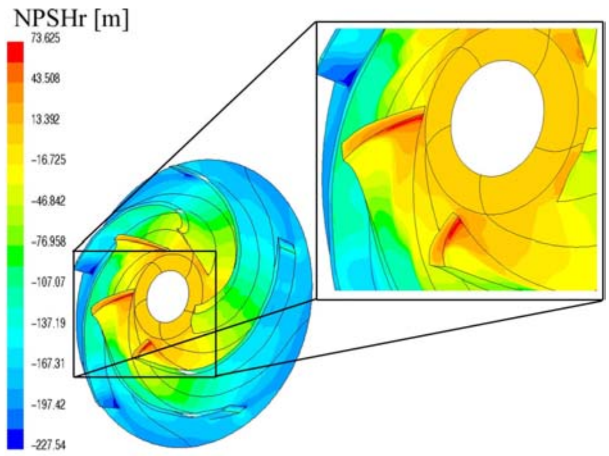

Figure 12. $NPSH_r$ distribution on the blades for mixed velocity distribution imposed on the inlet of the impeller

It is well known, the maximum risk of cavitation occurs where the largest $NPSH_r$ value (corresponding to minimum pressure) is obtained. Due to the sharpness of the leading edge, the maximum $NPSH_r$ value appears at the junction between blade suction side and the crown near to the leading edge (the red spots in Figures 11 and 12). According to expectation, a largest cavitation risk is obtained for non-uniform velocity field relative to the uniform one. This remark can be qualitatively observed in Figures 11 and 12 where the red spots on leading edge are larger for non-uniform velocity field then uniform one. Our numerical results are in good agreement with experimental observations, see Figure 13.

Both absolute and relative flow angles are defined as follow:

$$\alpha = arctg \frac{V_u}{\sqrt{V_r^2 + V_z^2}} \quad (7)$$

$$\beta = arctg \frac{\sqrt{V_r^2 + V_z^2}}{U - V_u} \quad (8)$$

The absolute and relative flow angles distributions on the outlet section of the pump suction are presented in Figures14 and 15. From the distribution of the absolute flow angle it can be observed that a large variation of this angle appears from the hub to the 50% span which leads to an unsteady flow.

Figure 13. Cavitation erosion on the impeller blades is displaced at the junction between suction side and crown. The numerical result are in good agreement with experimental observations.

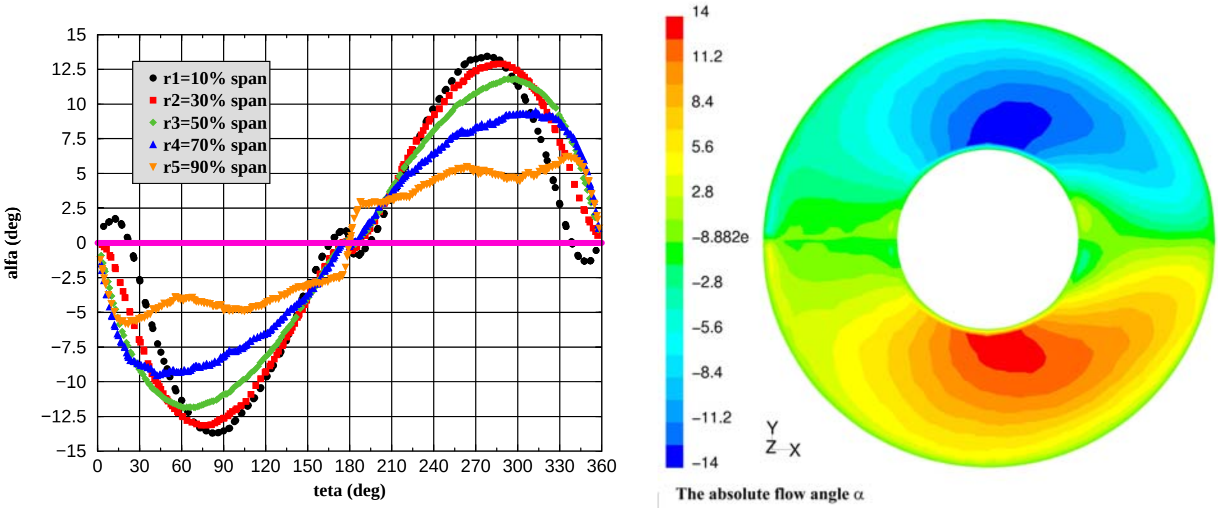

Figure 14. The absolute flow angle (α) distribution on the outlet section of the suction domain

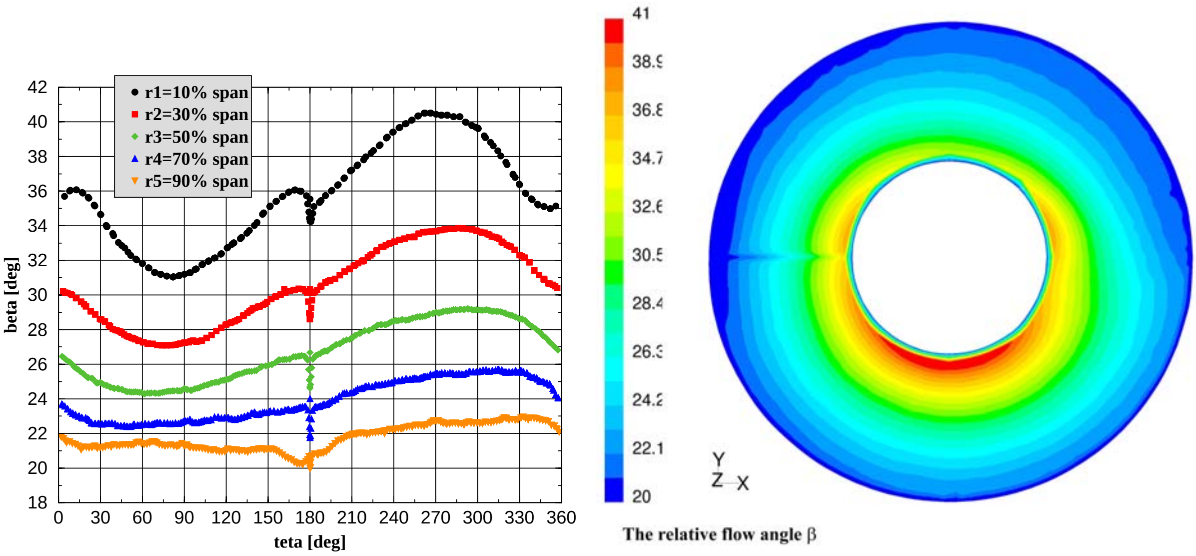

Figure 15. The relative flow angle (β) distribution on the outlet section of the suction domain

From the distribution of the relative flow angle one can observe that an important variation of the angle appears near the hub and that leads to cavitation in that area of the impeller.

4. CONCLUSIONS

The paper presents a numerical study of the 3D flow in the impeller of a storage pump. Comparison between numerical results computed with and without suction domain at best efficiency point is presented in order to evaluate the influence of the actual conditions. These results are important in the framework of hydraulic pump refurbishment, as well as for design optimization.

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 crown, in agreement with experimental observations.

The cavitation risk for the case of non-uniform velocity field imposed on the inlet of the impeller is larger than for uniform velocity field, respectively.

The further analysis of the flow in the storage pump impeller will take into consideration the unsteady flow.

ACKNOWLEDGMENTS

The present work has been supported from the National University Research Council Grant (CNCSIS) A-Consortium 33/2006. Numerical computations have been performed at the Numerical Simulation and Parallel Computing Laboratory from the “Politehnica” University of Timisoara, National Center 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. User's Guide*, Fluent Incorporated, 2001.
3. Fluent Inc., *Gambit 2. 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.