IOP Conf. Series: Journal of Physics: Conf. Series **813** (2017) 012048 doi:10.1088/1742-6596/813/1/012048

Decoupled CFD-based optimization of efficiency and cavitation performance of a double-suction pump

A Škerlavaj[1,2], M Morgut[2], D Jošt[1] and E Nobile[2]

[1] Kolektor Turboinštitut, Rovšnikova 7, 1210 Ljubljana, Slovenia
[2] University of Trieste, Department of Engineering and Architecture, Piazzale Europa 1, 34127 Trieste, Italy

E-mail: aljaz.skerlavaj@kolektor.com

Abstract. In this study the impeller geometry of a double-suction pump ensuring the best performances in terms of hydraulic efficiency and reluctance of cavitation is determined using an optimization strategy, which was driven by means of the modeFRONTIER optimization platform. The different impeller shapes (designs) are modified according to the optimization parameters and tested with a computational fluid dynamics (CFD) software, namely ANSYS CFX. The simulations are performed using a decoupled approach, where only the impeller domain region is numerically investigated for computational convenience. The flow losses in the volute are estimated on the base of the velocity distribution at the impeller outlet. The best designs are then validated considering the computationally more expensive full geometry CFD model. The overall results show that the proposed approach is suitable for quick impeller shape optimization.

1. Introduction

Centrifugal pumps are widely used in industrial applications. Double-suction (also called: double-entry) centrifugal pumps allow transportation of greater flow rates than single-entry pumps [1] because they are less prone to cavitation problems (smaller $NPSH_R$, required net positive suction head). Another advantage is counter-balancing of axial hydraulic forces due to double-entry design [1].

The optimization techniques bring many benefits over the traditional "trial-and-error" design process, including less hard-to-spot human-based errors. In turbomachines in general, usually multiple objectives are to be optimized. One of the first multi-objective optimization study was performed by Lipej and Poloni [2]. Due to importance of centrifugal pumps in industrial applications, as well as of pump-turbines, many recent studies aim to optimize their geometry to improve their performance. Shingai et al. [3] performed a multi-objective optimization of pump-turbine, where the objectives were efficiency of turbine, pump efficiency, cavitation characteristics for both types of operation, torque to hold runner blades and total pump head. Xuhe et al. [4] optimized pump turbine for efficiency in both modes of operation whereas Zhang et al. [5] performed optimization of a centrifugal pump for vibration, using fluid-structure interaction (FSI).

The purpose of this work is to propose and validate an economical decoupled simulation method for the optimization of a pump. In the presented decoupled optimization case, only the impeller geometry was allowed to be modified, while the rest of the geometry was fixed. Therefore, initially,

flow losses in the volute were estimated from CFD simulations as a function of the velocity distribution at the inlet of the volute. During the optimization only the flow in the impeller passage was simulated, whereas the flow losses in the volute were assessed from impeller outflow velocity distribution. The validation of the method was performed, for the most promising design candidates, by using a full geometry coupled method, considering the suction chambers, full impeller geometry and the double volute. However, given the interest in comparing the two methods, the non-uniformity of the flow at the impeller inlet due to the suction chambers was deliberately disregarded in case of the coupled method, by using the so-called stage (mixing plane) condition at the rotor-stator interface.

Besides the maximization of pump's hydraulic efficiency, also the minimization of the cavitation extent was set as the objective of the optimization. For the latter objective, a simple method for estimation of reluctance to cavitation was used and validated.

In this study, the specific speed nq of a pump was equal to 62 (specified per impeller side).

2. Numerical procedure

The geometry of the double-suction centrifugal pump is presented in figure 1. Instead of using the full-geometry (in this paper also called *coupled*) model (figure 1a), the optimization was performed for a *decoupled* model (figure 1c). The coupled model was used during the validation process, to compare the decoupled model to the coupled one.

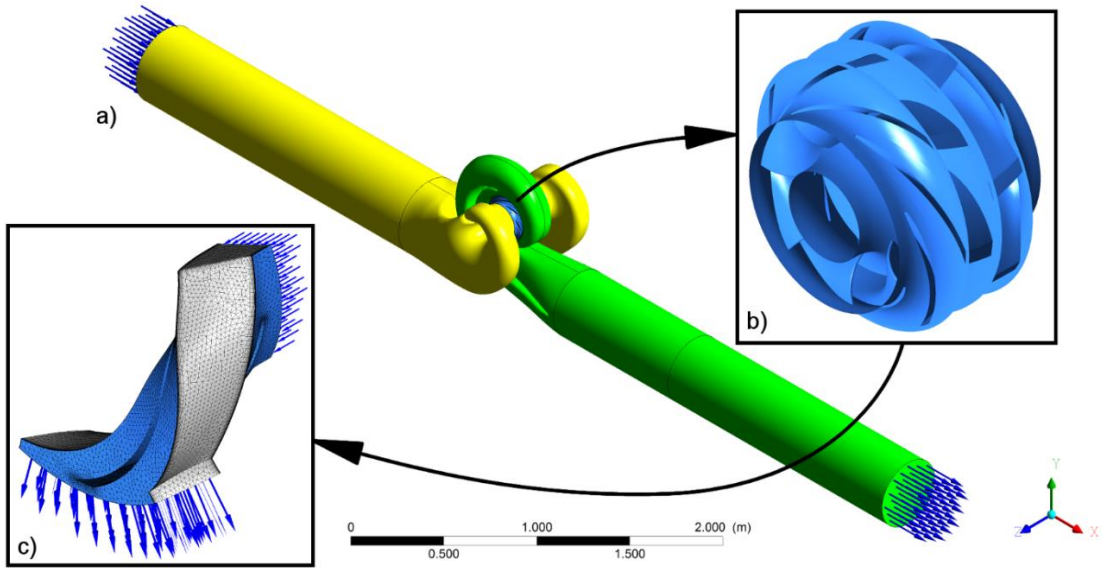

Figure 1. Double-suction centrifugal pump. (a) Geometry. In yellow: inlet pipe with suction chambers. In blue: impeller. In green: double volute with outlet pipe. (b) Double sided impeller. (c) Mesh of the impeller passage. Arrows represent inlet and outlet boundary conditions for the coupled (a) and the decoupled (c) simulation case.

It is well known that the suction chamber delivers a non-uniform flow to the pump impeller, which for instance results in length of the blade-attached cavities, varying upon the theta (circumferential) position of the impeller blade. In [6] it was shown that the growth and reduction of the cavities strongly depend upon the location, strength and rotation direction of the vortices formed in the suction chamber, close to the rib. In [7] the non-uniformity was quantified with the angle $\gamma = \mathrm{arctg}(v_{circ}/v_{axial})$.

Before performing the decoupled optimization, the coupled CFD simulation was performed, to obtain the velocity distribution at the impeller inlet. The angle γ and the velocity distribution at the grid interface between the suction chamber and the impeller is presented in figure 2.

In the optimization phase, which was performed with the decoupled CFD model (figure 1c), the velocity components at the inlet boundary condition were prescribed as a function of radius around the rotation axis (figure 2b). At the outlet, the average pressure boundary condition was set. Periodic boundary conditions (in grey in figure 1c) were used to take into account the periodicity of the impeller passage. For the impeller, the shape of the meridional channel was based on past experience and was not modified during the optimization procedure. On the other hand, the blade passage geometry was generated from a set of 14 input parameters provided by an in-house code that uses an inverse-singularity method. The data was exported to ANSYS DesignModeler commercial tool, where a 3-D geometry of an impeller was automatically created. For each geometry, a tetrahedral-based, unstructured mesh of the impeller was automatically created in ANSYS ICEM, which also provided the generation of 15 prism layers on the walls.

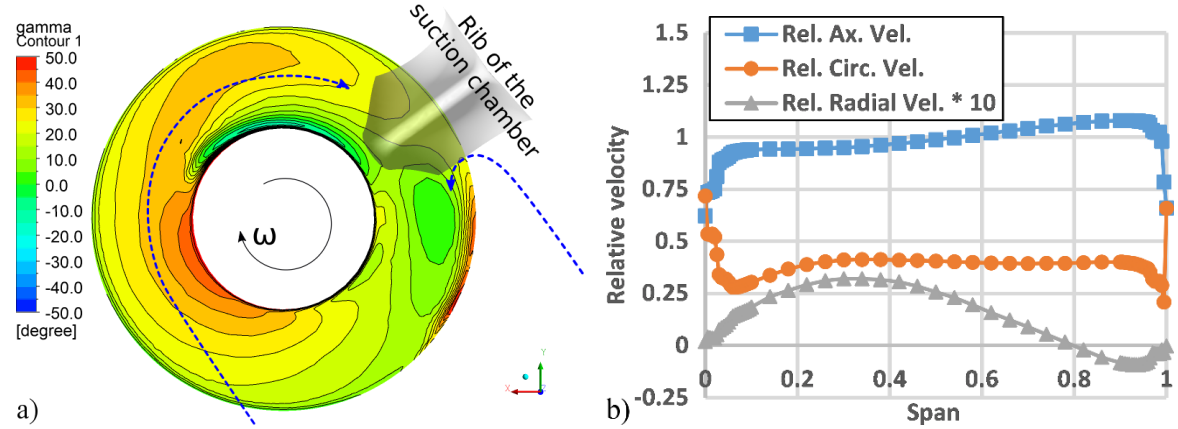

Figure 2. Flow conditions at the impeller entrance. (a) Contour plot of angle γ at the outlet from the asymmetric suction chamber. Blue dashed lines represent flow directions from the suction chamber. (b) Distribution of axial, circumferential and radial velocities at the entrance to the impeller. Velocities are relative to the average axial velocity at the entrance to the impeller. Radial velocity values in graph are multiplied by the factor of 10.

The optimization process was driven by modeFRONTIER 2014 [8] optimization platform. The objectives of the optimization were overall pump efficiency and reluctance to cavitation. The latter was assessed from the pressure difference between the impeller domain inlet boundary surface and the surface on the impeller blade, consisting of the leading edge and a part of the blade suction side. To obtain the overall pump efficiency, the flow losses of the volute ΔH_{volute} were assessed as a function of the angle α_2 between circumferential and absolute velocity at the impeller outlet, at regions near hub, at midspan and near shroud:

$$\Delta H_{volute} = \frac{f(\alpha_{2,hub})+f(\alpha_{2,midspan})+f(\alpha_{2,shroud})}{3}, \quad (1)$$

where the function f, a relationship between the flow losses and the α_2, was obtained prior to optimization from CFD simulations of the volute alone.

Besides the two objectives, two constraints were used: the overall pump head (as an interval), as it also influences the direction of the optimizer towards large heads [9, 10], and the allowed $\Delta\alpha_{2mid\text{-}sh}$, the difference of α_2 angle between the midspan and shroud intervals. Based on the volute simulations, the cases with the value of $\Delta\alpha_{2mid\text{-}sh}$ smaller than two degrees were considered as unfeasible.

Design of experiments (DOE), which fills the design space with the initial set of design variables, was based on the Latin Hypercube [11] samplings, but consisted also of some known high-efficiency designs, obtained in previous optimization cases. The Latin Hypercube sampling guarantees uniform random distribution of points over the variable range. The initial generation thus represented 48 sets of variables. Afterwards, the Multi Objective Genetic Algorithm (MOGA-II) [12], based on generational

evolution, was used for 10 generations. Probability of directional cross-over, probability of selection and probability of mutation were set to 0.5, 0.05 and 0.1, respectively.

The validation of the decoupled-simulation-based method was performed for the most promising design candidates, by using a full geometry coupled method. Namely, after performing CFD simulations on 480 cases, each one taking less than six minutes of clock-time to complete, the simulations were ordered by the predicted efficiency. The ones with the largest efficiency value were then validated using the full geometry model (figure 1a), using modeFRONTIER platform as a convenient way to drive the CFD simulations and performing the postprocessing.

Both, coupled and decoupled CFD simulations, were performed with the ANSYS CFX software. These were performed only for the design point, as steady-state single-phase simulations with Shear-Stress-Transport turbulence model [13] with a curvature correction [14] (SST-CC). The 'high-resolution' advection scheme was used, which is an upwind adaptive scheme, based on the Barth and Jespersen's limiter [15]. A local time-scale factor equal to 10 was used. For the *decoupled* case, simulations lasted for up to 550 iterations, or stopped when RMS residuals were smaller or equal to $1 \cdot 10^{-5}$. The result data (head, torque and efficiency) were averaged over the last 20 iterations to produce meaningful result. It was checked that even for simulations with shorter number of iterations the averaging sample size choice did not influence the value of the output variables. For the validation with the *coupled* case, the simulations were performed using the stage condition [16] as a general grid interface between the impeller and the suction chamber on one side, and the volute on the other side. In this case the CFD simulations lasted for 600 iterations. All other settings remained the same. The process of case generation, case simulation and its postprocessing lasted for about 2:45 hours, on 128 computer cores.

3. Results and discussion

Figure 3a presents predicted efficiency by using decoupled and full geometry numerical approach. Results are presented in relative form, ordered by the decreasing value of the efficiency of the decoupled simulations. Case 0 is the case with highest efficiency (relative efficiency of 100%). It can be noticed that the inclination of the trendline of the full geometry model prediction is similar to the inclination of the results of the decoupled approach, which means that in general such method can be used for the optimization. However, the line of predicted efficiency with full geometry approach is jagged, with values oscillating for approximately ±0.5%, so it is worth investigating the reasons for such behaviour.

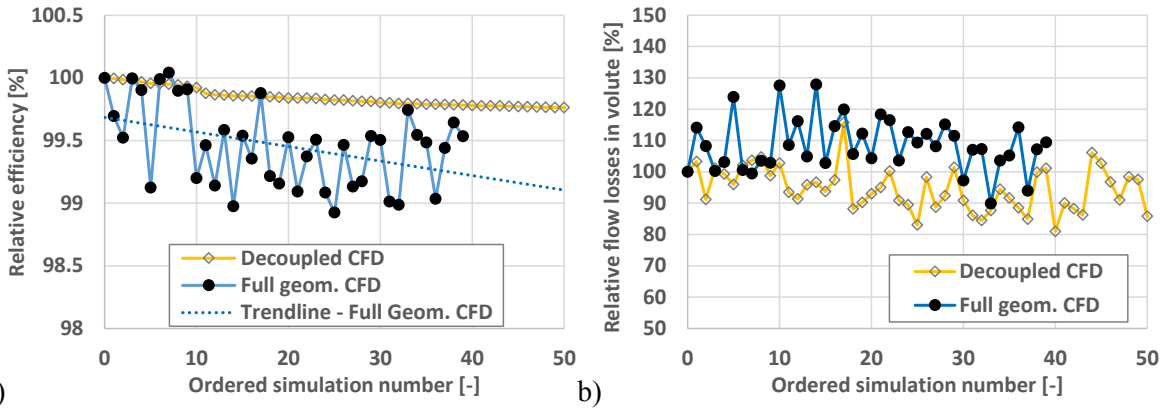

Figure 3. Comparison of (a) efficiency and (b) flow losses in the volute for decoupled and full-geometry steady-state simulations. Cases are ordered by efficiency, obtained by the decoupled numerical model. Efficiency and flow losses are relative to the values obtained in case 0.

Since for the decoupled approach the prediction of the flow losses in the volute is based on the α_2 parameter, it is worth comparing predicted values of α_2 at hub, midspan and shroud intervals for the two numerical approaches. In our case, for the decoupled approach, the α_2 at hub was overpredicted by approximately 1.3°, whereas the α_2 at shroud was underpredicted by approximately 1.3° (figure 4). At midspan, the underprediction was negligible, 0.2°. The systematic deviations at hub and at shroud, due to the volute interaction with the impeller, can be taken into account in future optimization processes, to improve the results. When the systematic deviations in figure 4 are eliminated, the discrepancy between the two curves in each graph is small, usually smaller than 1°. Nevertheless, flow losses, predicted by equation (1), do not change much when the corrected value of α_2 is used.

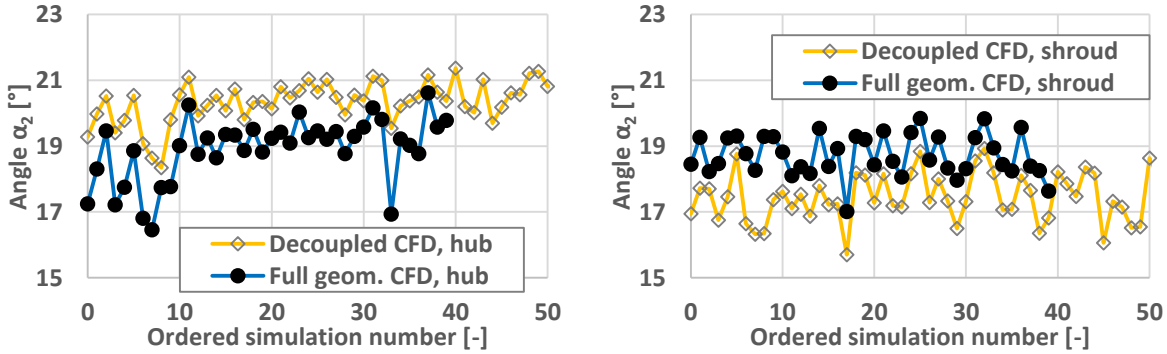

Figure 4. Angle α_2 at hub and shroud, predicted by decoupled and full-geometry steady-state simulations. The order of the cases is the same as in figure 3.

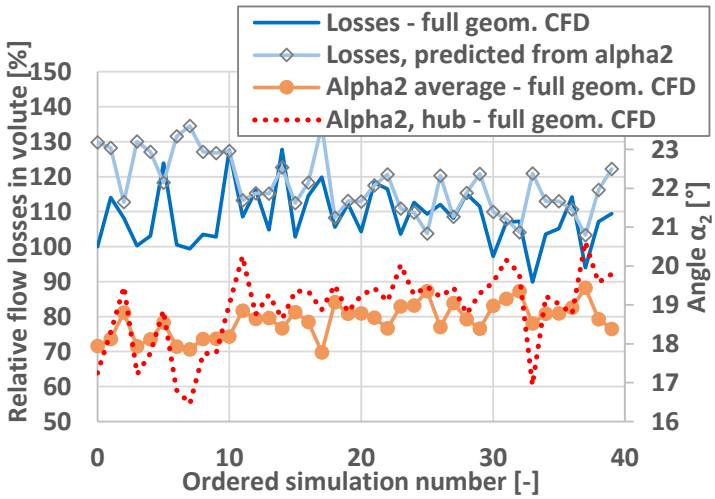

Figure 5. Primary axis: relative (to case 0 baseline, as predicted by the CFD) flow losses in volute for the full geometry model - comparison of CFD and prediction by equation (1). Secondary axis: angle α_2 - average value and value at hub. Order of cases is the same as in figure 3.

For the reason presented above it is necessary to consider a modification of equation (1), to address other reasons influencing the volute flow losses. In figure 5 only the full geometry (coupled) CFD model is considered. For the full model, the flow losses predicted by equation (1), where angles α_2 are obtained from the coupled simulations, are compared to the "real" losses (blue curves). It can be noticed that the approach presented by equation (1) is acceptable for designs from case 10 onwards. However, large discrepancy can be noticed for cases 0, 3, 4, 6, 7, 8, 9 and 33. These cases seem to be the cases where the value of $\alpha_{2,hub}$ is smaller than the average value of α_2 (figure 5). Small value of

$\alpha_{2,hub}$ seems to act beneficially on reducing the flow losses in the volute, so the equation (1) could be modified by using a weight factor, accounting for the low value of $\alpha_{2,hub}$.

Apart from the efficiency, another objective of the optimization was reluctance to cavitation. A very simple variable was considered as indicative of this risk: minimal value of pressure difference Δp between the impeller domain inlet boundary surface and the surface on the impeller blade, consisting of the leading edge and a part of the blade suction side. In figure 6 it is possible to observe that the decoupled and the full geometry CFD approach predicted similar values of Δp. In figure 7 the values of Δp can be observed for cases 14 and 33. Although only the minimal value of such pressure difference is presented in figure 6, it can be observed in figure 7 that low-pressure values are attributed to a part of a surface, rather than just one point. This means that the result is not just a pure coincidence, based on a location of a node of the surface mesh. The proposed variable is therefore a good simple candidate for assessment of reluctance to cavitation.

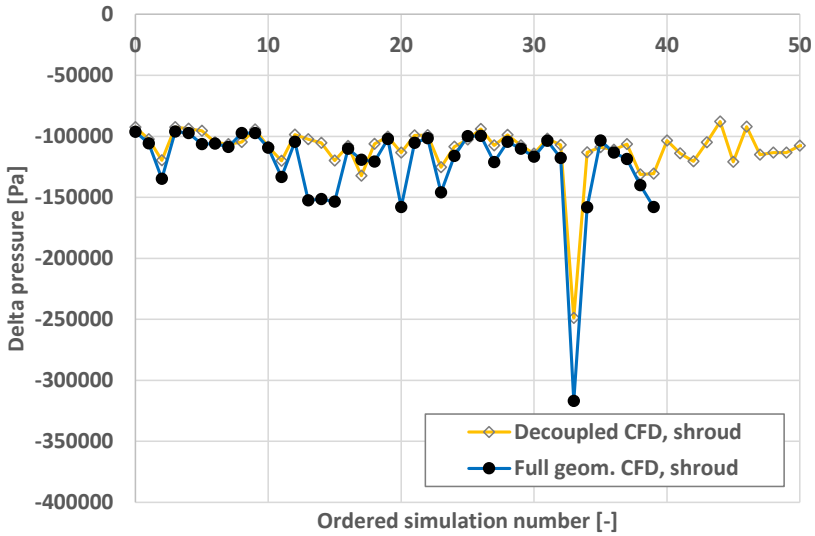

Figure 6. Pressure difference between pressure at inlet to impeller domain and pressure at blade surface (leading edge and suction side). Comparison of decoupled and full-geometry steady-state CFD simulations. Order of cases is the same as in figure 3.

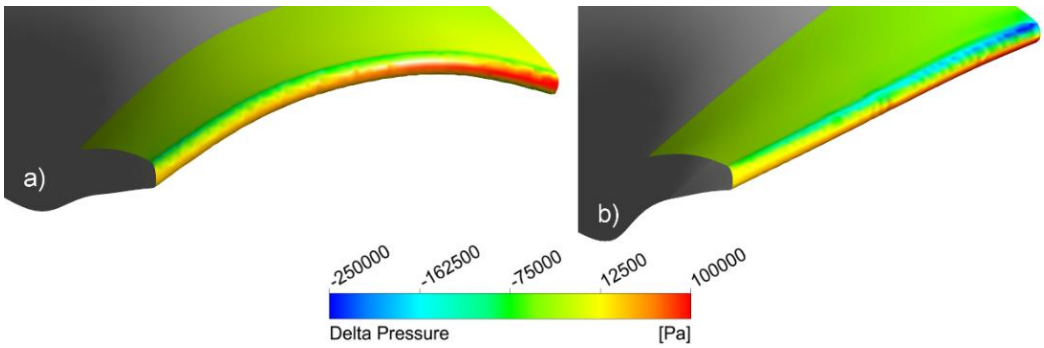

Figure 7. Pressure difference, predicted by decoupled approach: (a) case 14; (b) case 33.

4. Conclusions

A decoupled numerical model for the optimization of a double suction pump's impeller was presented. Results for the most promising candidates for high pump efficiency were validated with the full geometry CFD numerical models. The following conclusions were drawn:

- The presented method with estimation of flow losses in the volute represents a good approach for quick optimization;

- Values of angle α_2 at hub and at shroud predicted with the decoupled approach need small adjustments;
- Low value of angle α_2, when compared to the average α_2 value, acts beneficially on reducing the volute flow losses.
- The value α_2 at hub should be included in the equation for determination of the volute flow losses;
- The presented Δp variable is a good simple candidate for the assessment of reluctance to cavitation.

Acknowledgements
The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n°612279 and from the Slovenian Research Agency ARRS - Contract No. 1000-15-0263.

References
[1] Gülich JF 2008 *Centrifugal Pumps* (Berlin: Springer-Verlag) p 926
[2] Lipej A and Poloni C 2000 Design of Kaplan runner using multiobjective genetic algorithm optimization *J. Hydraul. Res.* **38**(4) pp 73–9
[3] Shingai K, Omura Y, Tani K, Tominaga H and Ito S 2008 Multi-objective optimization of diagonal flow type reversible pump-turbines *Proc. of 24th Symp. on Hydraulic Machinery and Systems* October 27-31 Foz do Iguassu (Madrid: IAHR)
[4] Xuhe W, Baoshan Z, Lei T, Jie Z and Shuliang C 2014 Development of a pump-turbine runner based on multiobjective optimization *IOP Conf. Ser.: Earth Environ. Sci.* **22** 012028
[5] Zhang Y, Hu S, Zhang Y, Chen L 2014 Optimization and analysis of centrifugal pump considering fluid-structure interaction *Scientific World J.* **2014** 131802
[6] Škerlavaj A and Pavlin R 2014 Effect of vortical structures on cavitation on impeller blades in pumps with suction chambers *IOP Conf. Ser.: Earth Environ. Sci.* **22** 052002
[7] Muntean, S., Škerlavaj, A., Drăghici, I. and Anton, L.E., 2015. Numerical analysis of the flow non-uniformity generated by symmetrical suction elbows of the large storage pumps *Proceedings of 6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, Ljubljana, Slovenia.
[8] ESTECO 1999-2016 *modeFRONTIER User Manual* (ESTECO: Trieste)p 504
[9] Kim JH, Oh KT, Pyun KB, Kim CK, Choi YS and Yoon JY 2012 Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics *IOP Conf. Ser.: Earth Environ. Sci.* **15** 032025
[10] Škerlavaj A, Morgut M, Jošt D and Nobile E 2016 Optimization of a single-stage double-suction centrifugal pump *To appear in proc. of 34th UIT Heat Transfer Conference* July 4-6 Ferrara
[11] McKay MD, Conover WJ and Beckman RJ 1979 Latin hypercube sampling: a comparison of three methods for selecting values of input variables in the analysis of output from a computer code *Technometrics* **21** 239–45
[12] Poles S 2003 *MOGA-II An improved Multi-Objective Genetic Algorithm* (Trieste: ESTECO)
[13] Menter FR, Kuntz M and Langtry R 2003 Ten years of industrial experience with the SST turbulence model *Turbulence, Heat and Mass Transfer 4* ed K Hanjalić, Y Nagano et al (New York: Begell House) pp 625-32
[14] Smirnov PE and Menter F 2009 Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term *J. Turbomach.* **131** 041010
[15] Barth TJ and Jespersen DC 1989 The design and application of upwind schemes on unstructured meshes *Proc. of 27th Aerosp. Sci. Meet.* (Reno: AIAA) 0366 (12pp)
[16] ANSYS 2016 *ANSYS CFX-Solver Theory Guide* (ANSYS: Canonsburg) p 362