PUMP SUCTION SHAPE OPTIMIZATION USING A PARALLEL STOCHASTIC RADIAL BASIS FUNCTION METHOD

T. KRATKY[1], L. ZAVADIL[1] & M. TABATABAEI[2]
[1]Hydraulics Research Centre, Ltd, Lutín, Czech Republic.
[2]University of Jyväskylä, Jyväskylä, Finland.

ABSTRACT
This paper deals with a shape optimization of pump suction, with the objective of improving the pump performance. A combination of ANSYS CFX software tools and a surrogate-based, so-called multistart local metric stochastic RBF (MLMSRBF) method for the global optimization of "expensive black-box functions" is employed. The shape of the suction is driven by 18 geometric parameters, and the cost functional is based on the CFD results. The practical aspects of assembling and evaluating a parametric CFD model, including mesh independence study, are shown. After initial design of experiment evaluation, a response surface model is created and used for generating new sample points for the expensive CFD evaluation. Then, the whole process is repeated as long as necessary. A parallel version of the method is used, with necessary modifications for dealing with CFD-specific problems, such as failed designs and uncertainty of computational times. Both steady-state and transient models are used for the optimization, each with a different objective function. The resulting designs are then compared with the original geometry, using a complete model of the pump and fully-transient simulation. Both hydraulic characteristics and multiphase cavitational simulations are considered for the comparison. At the end, the results and challenges of using these methods for CFD-driven shape optimization are discussed.
Keywords: CFD, parallel optimization, shape optimization, stochastic RBF, surrogate-based.

1 INTRODUCTION

Employing CFD simulations for parametric studies and shape optimization is currently a standard part of computer-aided hydraulic design. The "usual" approach uses Design-of-Experiment (DOE), response surface, sensitivity analysis, and possibly gradient-based optimization. While this routine usually works sufficiently well, it still requires a relatively high number of the expensive CFD computations – easily exceeding one hundred.

Some publications propose advanced surrogate-based methods for the global optimization of expensive "black-box" functions. Considering the computational expenses of CFD simulations of pumps, it was decided to test one such method, namely, the Stochastic RBF method, described in Regis and Shoemaker [1, 2].

2 ORIGINAL PUMP AND OPTIMIZATION GOALS

The pump is radial with horizontally mounted shaft and spiral casing; shrouded impeller has six blades. The specific speed was $ns = 135$. To improve the suction performance, an inducer was developed. The hydraulic design of the inducer was based on Gulich [3]. For CFD analysis, a complete model of the pump hydraulic parts was assembled and computed using ANSYS CFX (Fig. 1). It included suction part, rotating inducer and impeller and stator part. The inflow and outflow sections were modified to improve the numerical stability of the simulation.

The meshes were a combination of structured hexahedral and tetra with prism. The CFD model was considered as fully-transient, with rotating inducer and impeller and rotor-stator interfaces. Two phases (water and water vapour) were considered and k-ω SST model was

ISSN: 2046-0546 (paper format), ISSN: 2046-0554 (online), http://www.witpress.com/journals
DOI: 10.2495/CMEM-V5-N5-667-677

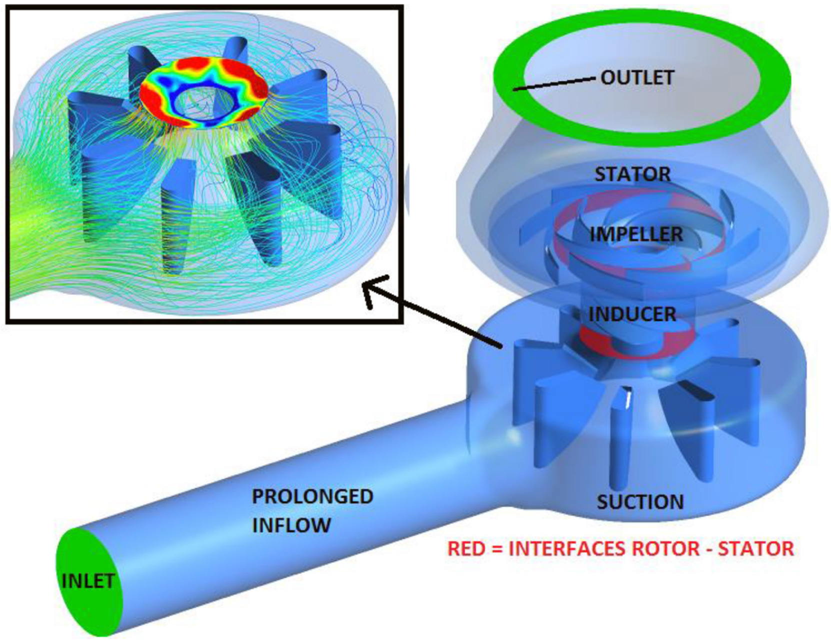

Figure 1: CFD model and details of the CFD results for the suction. Colours at the interface show differences from the average velocity. Red means +10% (and more), blue -10%.

used for turbulence modelling. Pressure at the inlet and mass flow at the outlet were prescribed for the boundary conditions. Next, a full set of CFD computations, including $NPSH_3$, was performed.

As the CFD analysis has shown, the uniformity of velocity field at the inlet part of the inducer was not very good. Since the suction part can be computed as stationary and thus relatively fast, it was decided to be a good start for implementing and testing a new optimization method. Two goals were considered – minimizing the pressure losses of the inducer and making the velocity field at the suction outlet as homogenous as possible. To limit computational expenses, only the optimal flow rate was considered.

3 OPTIMIZATION

3.1 Parametric model

For the purposes of the optimization, only the suction part was considered. The geometry was created using ANSYS DesignModeler. Hydraulic design was based on the results of the original suction and general tips from Gulich [3]. In total, there were 18 geometric parameters - outer wall diameter, blade length and various radii (Fig. 2). The blades were considered to be all the same. The inflow and outflow parts of the suction were modified for better numerical stability of the CFD simulation. The mesh was generated as a tetra/prism in ANSYS Meshing.

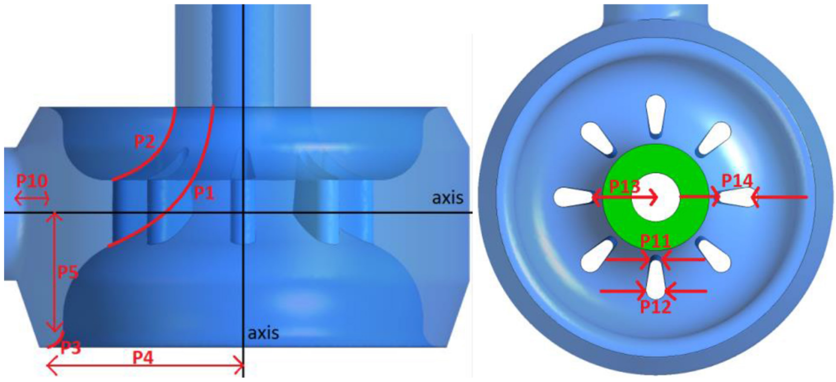

Figure 2: Visualization of selected parameters, defining the suction geometric shape.

The CFX model was set as a single-phase (water) and steady state, the boundary conditions and turbulence model remained the same as for the original fully-transient model. The model was created as ANSYS Workbench project, but CFX definition files creations, computations and results evaluation were done in batch mode, employing custom Python codes and Linux shell scripts. This gave more flexibility than ANSYS Workbench. It was also much easier to generate the geometries and meshes for different parameter combinations and check errors.

3.2 Optimization method

In accordance with general recommendations in Tabatabaei *et al*. [4], a surrogate-based method was selected, namely the parallel version of stochastic RBF described in Regis and Shoemaker [2]. This method has a convergence guarantee to the global optimal solution in a probabilistic manner. It works as follows:

1. Generate initial DOE and compute the samples.
2. Use already computed samples and fit the response surface.
3. Generate randomly candidate points (ca. 10,000) to cover the response surface uniformly.
4. Select N new points for evaluation, as a compromise between exploiting local minima of the response surface and exploring areas further away from the already sampled points.
5. Evaluate the selected points.
6. Repeat (2) until ending criteria are met.

N can be selected arbitrarily, the authors recommend either 4 or 8 for optimal performance. The response surface approximation is based on radial based function (RBF). The details are described in Regis and Shoemaker [1, 2], and the authors suggest this as the best option, especially for higher number of parameters.

Freely available Matlab codes were used and modified for usage with CFD simulations. The original code expects to obtain results for each black-box function call. But with CFD computations on the HPC cluster, there is resourcing problem. Geometry generation can fail for some parameter combinations. Computational times required for CFD can also differ a lot. They depend on scheduler system, CPU architecture and convergence speed. As the results, the objective function values are read in purely random order (or not at all). Of course, it is possible to guarantee results for any sample (by careful tuning of the parametric model + possibly “faking” the results of failed/delayed computations). But having an optimization code able to deal with these problems, is much more convenient.

Thus, the original method was modified to work with something that could be called “pipeline”.

1. Generate initial DOE and compute the samples.
2. Use already computed samples and fit the response surface.
3. Generate randomly candidate points (ca. ten thousand) to cover the response surface uniformly.
4. Select N new points for evaluation and add them to the queue.
5. *Wait until at least one computation finishes. Then wait for a few more minutes and read all available results. (Due to the scheduler system nature, the results often come within a very short interval.)*
6. *Select k new points, where k = number of samples computed + number of samples failed.*
7. *Add the newly selected samples to queue.*
8. Repeat (5) until ending criteria are met.

This way, the failed samples are simply ignored, and the “slower” ones are just used once they finish. Because of this, there are no bottlenecks caused by waiting for “stuck” computations.

3.3 Design of experiment

From the beginning, the initial response surface needs to be created. Latin hypercube sampling, included in the available Stochastic RBF codes, was used for the DOE table creation. For the 18 parameters, 36 samples were created and used as an input for the batch processing. 31 models were successfully generated and 5 failed because of geometry errors. Such a failure rate was considered to be acceptable. Thus, the CFD simulations could be performed.

The computations run on a Linux HPC cluster, using ANSYS CFX command line options. The number of iterations was not fixed, but “until reasonable convergence”. For this, Linux shell scripts and some Python codes were used. After some preliminary testing, the evaluated criteria were decided like these:

$$H_{\text{Suction}} = \frac{Total\ Pressure_{Inlet} - Total\ Pressure_{Outlet}}{\rho_{Water} \cdot g}$$

$$v_{diff} = \int_{Outlet} \left| v - v_{avg} \right|$$

Unfortunately, the computation proved to be challenging for the steady-state model. The process of water going through the suction is dynamic by its nature, even without the rotating inducer and impeller. As the result, the testing has shown too strong mesh dependence. Still, because testing optimization method with a steady-state CFD model is much less computationally demanding, it was decided to perform more thorough testing. For the whole DOE, four different meshes (ranging from ca. 100 thousand to 2 million nodes) were created, computed and evaluated. The details are described in Kratky *et al.* [5].

Based on the results, it was decided that for the $H_{Suction}$ criterion, the relative differences between various geometries remain similar enough for different meshes. The velocity uniformity v_{diff} proved to be too unreliable (Fig. 3). Thus, despite the original assumptions, the objective was changed to minimizing the $H_{Suction}$. Next, the best mesh setting was selected and the optimization was performed.

3.4 Optimization results

The results are typical for this method. In most cases, it is able to get an improvement from random search (DOE) in a few tens of samples. Considering the multiple sample computation, this usually means less than ten iterations. Further computations then do not yield any considerable progress (Fig. 4).

Next, for the best variants from both the DOE and the optimization, the full model was assembled. Then, performance and cavitational characteristics were computed (Fig. 5).

3.5 Parametric model with the inducer

As the next step, a different way of evaluating the velocity uniformity was chosen. The inducer was added to the model (Fig. 6), and it was considered as a transient case. Inflow and outflow parts were again modified due to numerical stability. The boundary conditions and turbulence model remained the same as in the previous case. Changing the computational model to transient increased computational demands considerably, but as a result, it allowed obtaining data for a meridional velocity profile at the inlet part of the inducer blades. Using ANSYS CFD-Post built-in functions, the Hub-to-Shroud line was defined

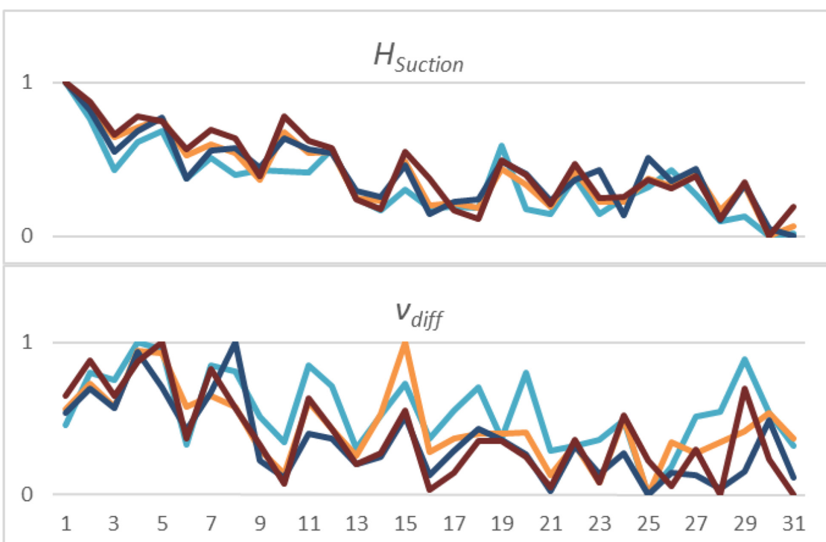

Figure 3: Results of DOE. To show the mesh influence on designs relative performance, the numbers are normalized to 0,1 interval for each mesh setting.

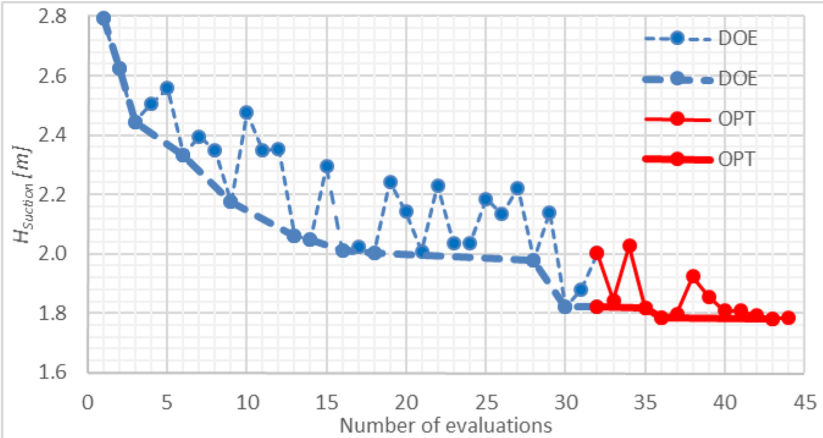

Figure 4: Objective function progress during optimization. The bold lines connect the best values reached.

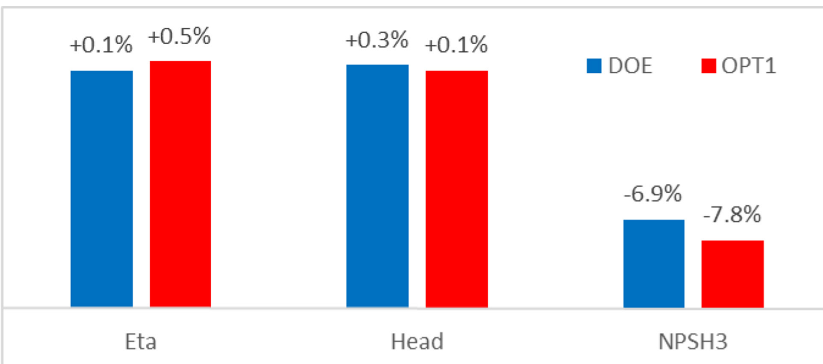

Figure 5: Hydraulic and cavitational performance of the optimized designs. The results are related to the original suction design.

and meridional velocity v_m was exported in multiple points. The objective function was then defined as:

$$J = \sum_{i=1}^{N} \frac{\left(v_m^{(i)} - \overline{v}_m\right)^2}{\overline{v}_m}$$

where $\overline{v}_m = \frac{1}{N} \cdot \sum_{i=1}^{N} v_m^{(i)}$ is the average meridional velocity on the selected Hub-to-Shroud line. Number of points N was selected as 32.

With an objective function defined like this, the optimization should yield a suction geometry ensuring as a uniform fluid entrance to the inducer as possible. Supposedly, this should lead to better pressure distribution along the blades and slower creation of the cavitational areas.

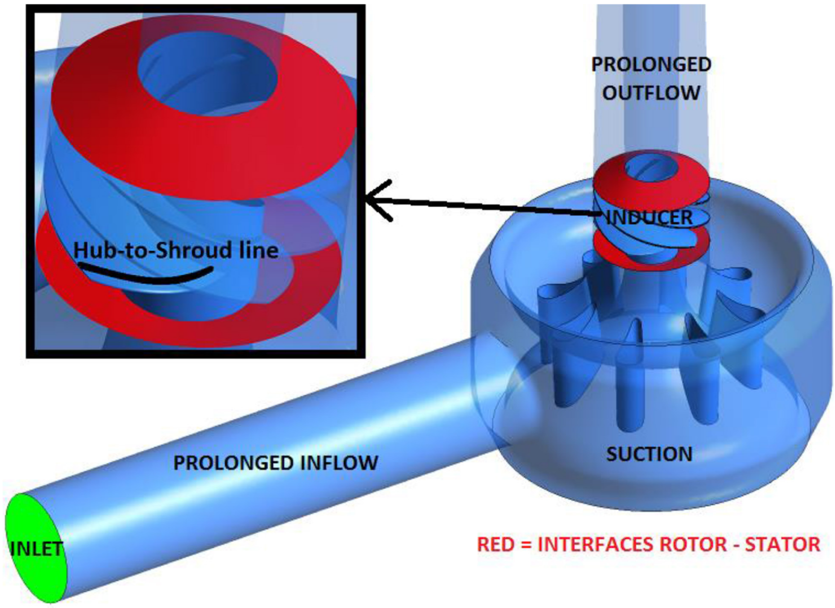

Figure 6: Parametric CFD model with the inducer and a detail of the Hub-to-Shroud line.

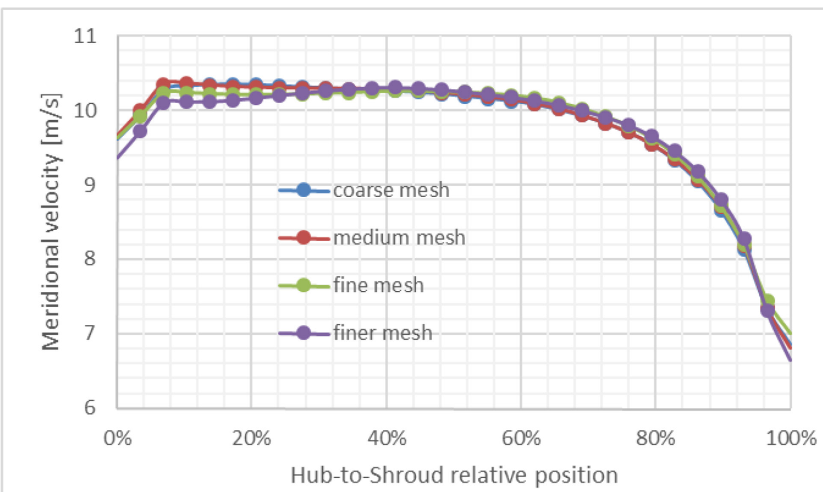

Figure 7: Meridional velocity profiles on the selected Hub-to-Shroud line.

The mesh dependence was again tested with the four different suction mesh settings (Fig. 7), this time only with one selected design. The inducer mesh remained the same for all the variants.

The transient analysis is more accurate, as expected. Once again, the geometries for all the samples from DOE were generated and computed on an HPC cluster. Then, started from these results, the Stochastic RBF optimization was performed (Fig. 8).

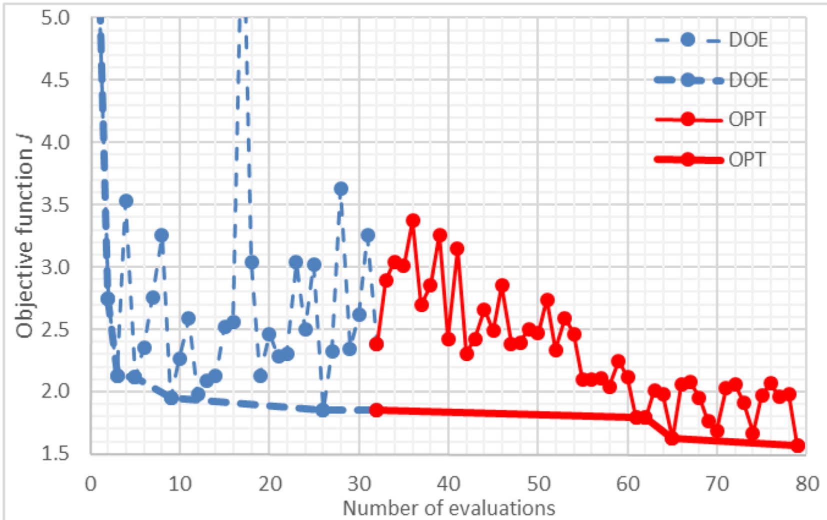

Figure 8: Objective function progress during optimization. The bold lines connect the best values reached.

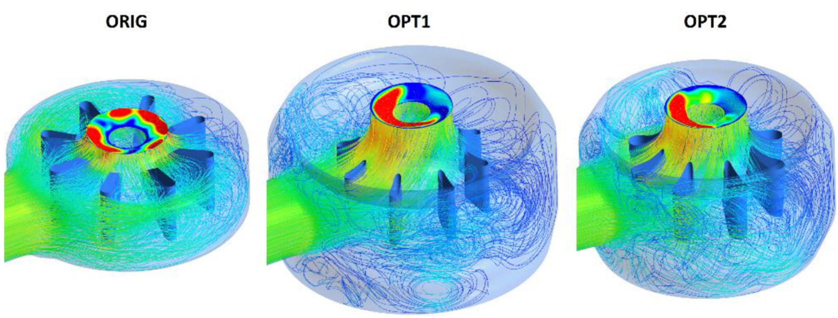

Figure 9: Suction designs and the results of CFD analyses. The colours show differences in velocity profile at the suction outlet.

3.6 Results of the optimization with the inducer

Quite unexpectedly, this objective function yielded inferior results to the pressure drop version (Fig. 9 and 10). Further analysis of the results has shown that the suction shape has very little to no effect on the velocity profile in the middle and trailing parts of the inducer blades. Due to this, the shape optimization with respect to the meridional velocity uniformity has no effect once the cavitational areas start to develop. Lowering suction pressure drop in the suction, on the other hand, gives more NPSH reserve.

3.7 Mesh dependence tests

To judge the accuracy of the numerical simulations, the geometry from the first optimization (with respect to the $H_{Suction}$) was tested on different meshes. The full set of both hydraulic and cavitation simulations was performed (Fig. 11). Three new variants were prepared – one with

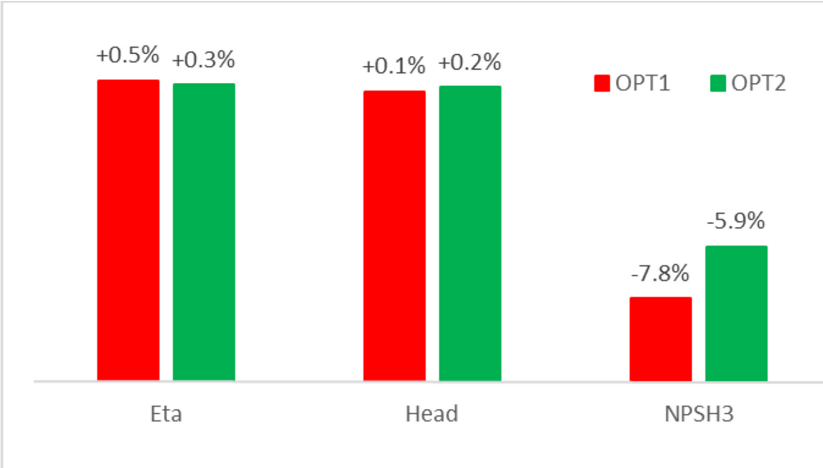

Figure 10: Hydraulic and cavitational performance of the optimized designs. The results are related to the original suction design.

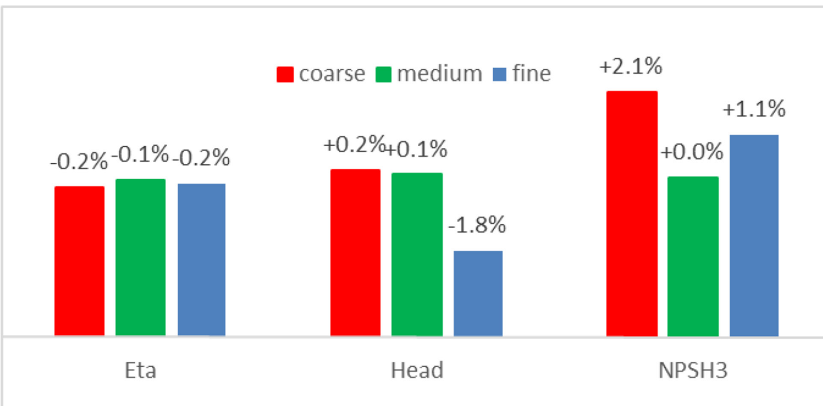

Figure 11: Hydraulic and cavitational performance mesh dependence. The results are related to the original mesh.

coarse mesh, one with similar node count as the original one, but with different topology, and one refined. The node count ranged from 1.3 to 7 million, compared to the 1.9 million for the original mesh.

The results vary slightly for different meshes. Usually, this is caused by different mesh sizing on the inlet and outlet part.

Overall, the results of the numerical simulations suggest the model with suction optimized for $H_{Suction}$ shows both the best efficiency and cavitational properties. Unfortunately, for hydraulic parameters the differences are too small to rule out the mesh-related errors. For the $NPSH_3$ characteristics, on the other hand, the changes are more substantial. Also, for this particular pump, the head drop is very steep, as can be seen in Fig. 12. This leaves the smaller range of possible errors and has a positive effect on $NPSH_3$ computation accuracy.

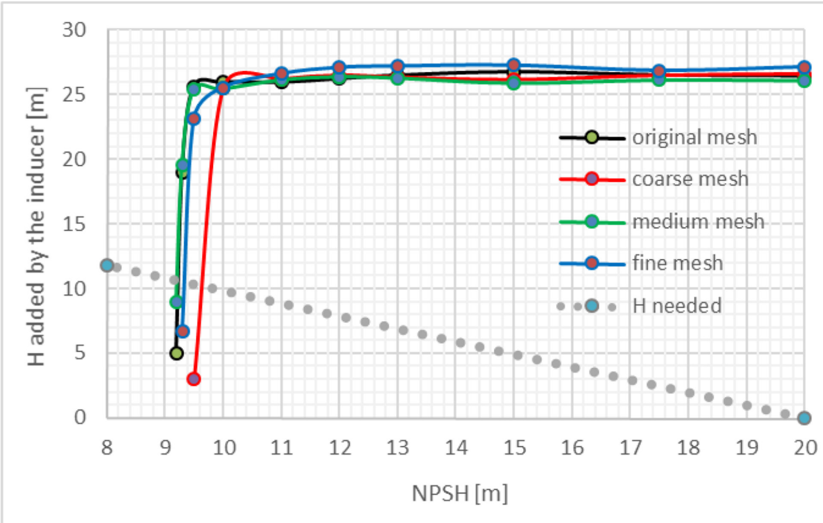

Figure 12: Head-drop curve mesh dependence. The dotted line shows the head boost required by the impeller to work properly. Once the static pressure generated by the inducer drops below this value, the cavitational breakdown of the pump occurs.

4 CONCLUSION

The selected method proved to be very efficient for CFD-driven shape optimization. Even for 18 parameters, the method can both improve the objective function relatively fast and deal with failed samples.

In this particular case, the steady-state model yielded some results, considering the computational expenses. Especially the cavitational performance was improved decently. For the transient case, on the other hand, optimizing the suction together with the inducer would be probably a better option. Overall, getting more decisive improvements would require optimizing different hydraulic parts of the pump. While the influence of the suction shape on the pump performance can be observed, it is comparable in magnitude to CFD numerical errors and thus problematic to optimize due to CFD accuracy.

Also, a multiobjective optimization needs to be utilized for the future cases. While the Stochastic RBF method is not designed for finding Pareto optimum, it can still be handled using scalarization approach described in Miettinen *et al.* [6].

ACKNOWLEDGEMENTS

The work was performed with support of TA ČR project TE02000232 Special Rotary Machines Engineering Centre.

Computational resources were provided by the CESNET LM2015042 and the CERIT Scientific Cloud LM2015085, provided under the programme “Projects of Large Research, Development, and Innovations Infrastructures”.

The authors thank to to internal IGA grant of the Palacký University (IGA_PrF_2016_022).

REFERENCES

[1] Regis, R.G. & Shoemaker, C.A., A stochastic radial basis function method for the global optimization of expensive functions. *INFORMS Journal on Computing*, **19**(4), pp. 497–509, 2007.
http://dx.doi.org/10.1287/ijoc.1060.0182

[2] Regis, R.G. & Shoemaker, C.A., Parallel stochastic global optimization using radial basis functions. *INFORMS Journal on Computing*, **21**(3), pp. 411–426, 2009.
http://dx.doi.org/10.1287/ijoc.1090.0325

[3] Gulich, J.H., *Centrifugal Pumps*, 2nd edn., Springer-Verlag: Berlin Heidelberg, 2008.

[4] Tabatabaei, M., Hakanen, J., Hartikainen, M., Miettinen, K. & Sindhya, K., A survey on handling computationally expensive multiobjective optimization problems using surrogates: non-nature inspired methods. *Structural and Multidisciplinary Optimization*, **52**(1), pp. 1–25, 2015.
http://dx.doi.org/10.1007/s00158-015-1226-z

[5] Kratky, T., Zavadil, L. & Bartonek, L., A parametric model of a pump suction. *Proceeding of the 24th SVSFEM ANSYS Users' Group Meeting and Conference 2016*, Dolni Morava, Czech Republic, pp. 111–119, 2016.

[6] Miettinen, K., Ruiz, F. & Wierzbicki, A.P., *Introduction to Multiobjective Optimization: Interactive Approaches*. Springer-Verlag: Berlin Heidelberg, 2008.