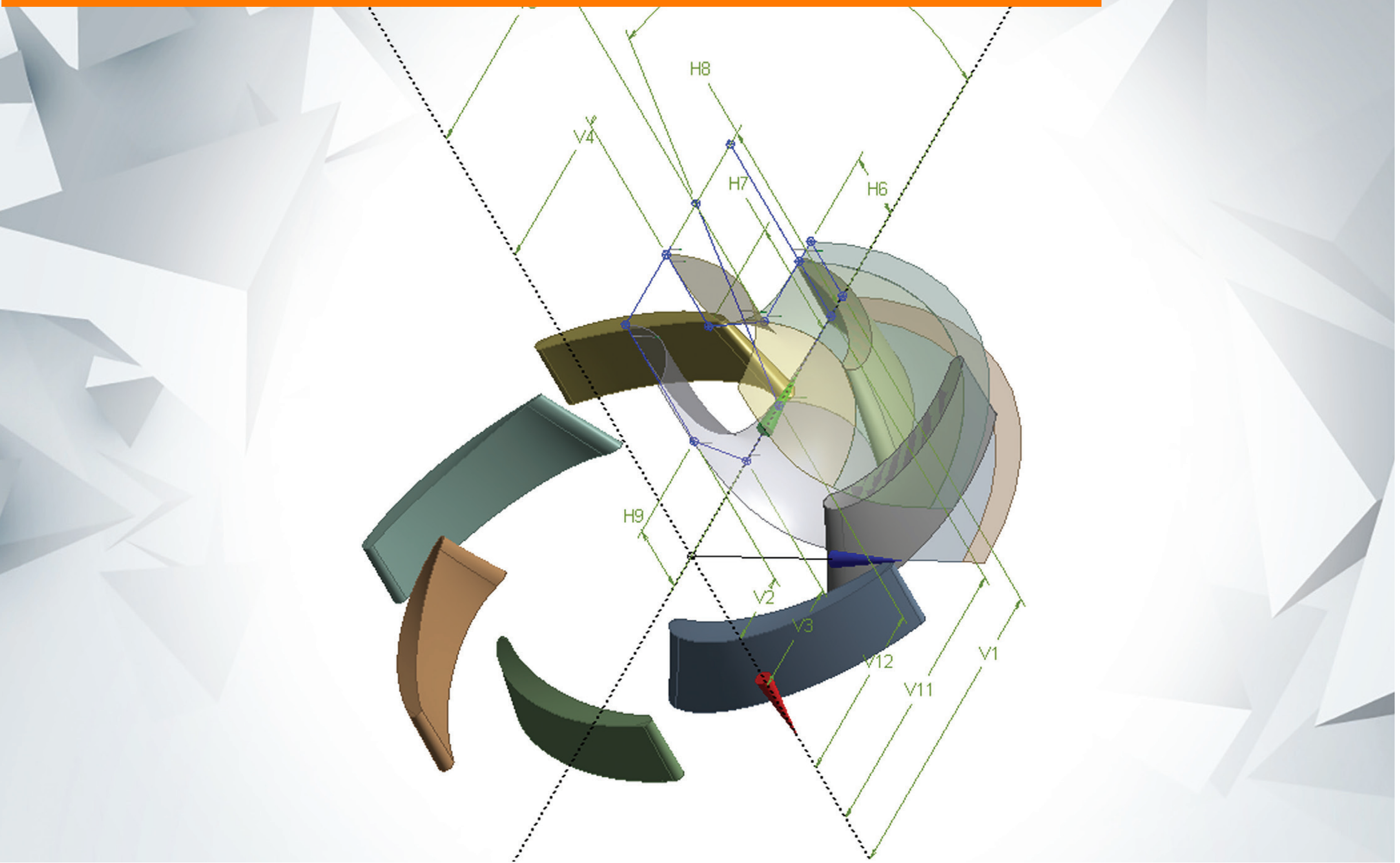
published in ARGE Dergisi 2019 |
Source: www.dynardo.de/en/library



Alev Elikalfa
alev.elikalfa@figes.com.tr
CFD Uygulama Mühendisi
FİGES A.Ş.

Dr. Ertan Karaismail
ertan.karaismail@figes.com.tr
Akışkanlar Mekaniği ve Termodinamik
Birim Yöneticisi
FİGES A.Ş.

DESIGN AND OPTIMIZATION OF A CENTRIFUGAL PUMP IMPELLER WITH ANSYS - optiSLang

1. INTRODUCTION

Pumps are the most widely used rotodynamic machines in fluid works. Depending on their types and areas of use, they can be driven by an electric motor, an engine or a turbine. In any case, they require external energy to operate. Considering the fact that most of them run continuously, especially in industrial applications, it would not be wrong to describe them as big energy consumers.

Among different types, centrifugal pumps are the most widely used ones [1]. Its performance depends on the system resistance. It is desirable to run the centrifugal pumps at or close to their Best Efficiency Point (BEP). However, due to varying operational conditions, this may not be always possible, and hence, preferred or allowed operating regimes are considered. What makes a pump great is a high efficiency at the rated conditions and slight performance variation (i.e. wide preferred or allowed operating regime) with increasing system resistance. Design of an efficient pump requires in depth understanding of the fluid flow through the pump and the effects of design parameters on the performance. Impellers can be considered as the heart of the pumps, since most of the pressure loss occurs here in a pump. In overall, optimization of impellers is crucial for an efficient pump design.

Computational Fluid Dynamics (CFD) simulation coupled with smart sensitivity and optimization algorithms is the most satisfying and advanced design approach not only for pumps but also for other fluid machines. Automatic optimization is crucial in reducing development time and costs too.

The basic steps of this approach can be summarized as

- Generation of an initial geometry for a given design requirement such as flow rate and head at the rated conditions, via one-dimensional (1-D) semi-empirical calculations.
- Conversion of geometric model into a parametric model that defines the shape and enables geometric modifications.
- Performing three-dimensional (3-D) CFD simulations.
- Performing sensitivity analysis and optimization to find the set of parameters for the best efficiency.

A sensitivity analysis is applied to detect and separate the design parameters which have no or only minor influence on the pump performance and efficiency. [2] After performing the sensitivity analysis, optimization is carried out in order to obtain the set of design parameters that will result in highest hydraulic efficiency. Since the optimization algorithm filters out the irrelevant parameters, the number of simulations to be performed is significantly reduced.

If a scatter in material properties, geometry, process or operational conditions exists and is deemed to significantly affect the performance of important response values or there is a doubt that safety distances may be too small or large, robustness analysis is applied. [3]

In this study, a centrifugal pump impeller was designed and optimized by utilizing ANSYS CFX and turbo tools along with ANSYS optiSLang module as for the Metamodel Based Optimization (MOP) based optimization for maximized hydraulic efficiency and minimized torque.

2. METHODOLOGY

A centrifugal pump impeller is to be designed and optimized to provide a flow rate of 25 kg/s at 350 kPa when running at a rotational speed of 2,000 rpm. The hydraulic efficiency of the impeller is aimed to be maximized.

The first step of the design and optimization process is to setup the workflow which consists of meanline design, geometry preparation, meshing, CFD simulations and an optimization loop that manages the process with respect to the CFD simulation outcomes. Figure 1 shows the ANSYS workflow prepared in ANSYS-Workbench. Here, ANSYS Vista CPD is used to obtain the meanline design, BladeGen is used to create geometry, Design Modeler is used to parameterize the blade geometry and ANSYS TurboGrid is used for grid generation. CFD simulations are carried out using ANSYS CFX. OptiSLang is used to manage the process by changing the design parameters in a smart way and to find of the set of parameters for the best efficiency.

2.1 Geometric Model

The base impeller is designed using ANSYS Vista CPD and BladeGen tools. The head, volumetric flowrate, rotational speed and other design parameters, shown in Figure 2, are required in order to obtain the initial impeller design in Vista CPD. The blade shapes, generated by Vista CPD has no blade thickness. A thickness distribution along the chord of the blade is applied in ANSYS BladeGen module.

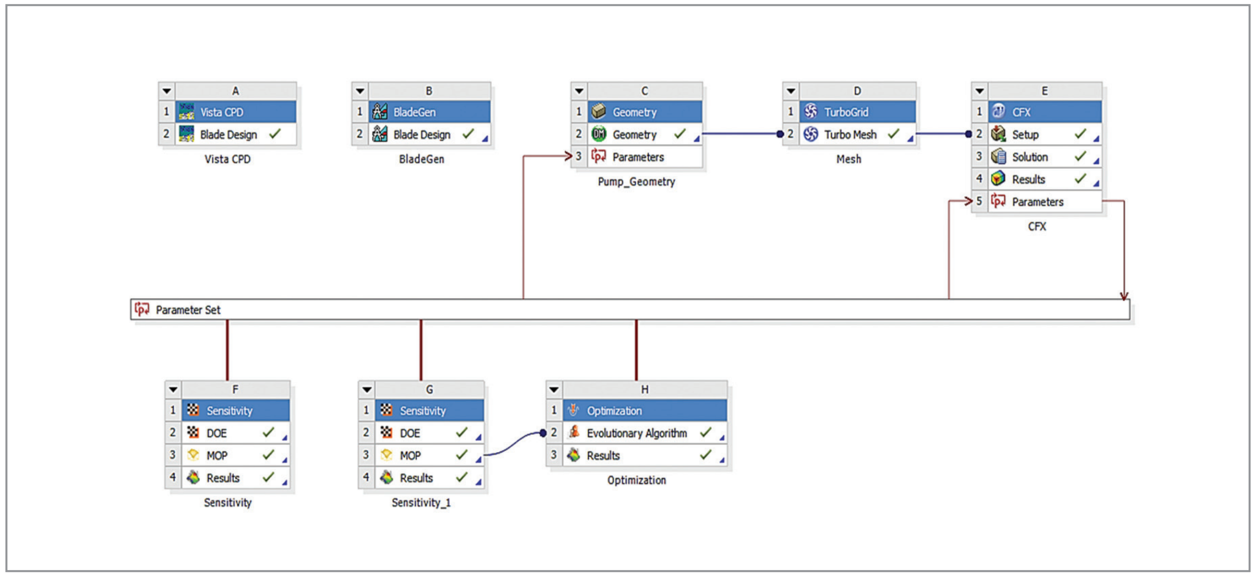


Figure 1. Optimization process flowchart

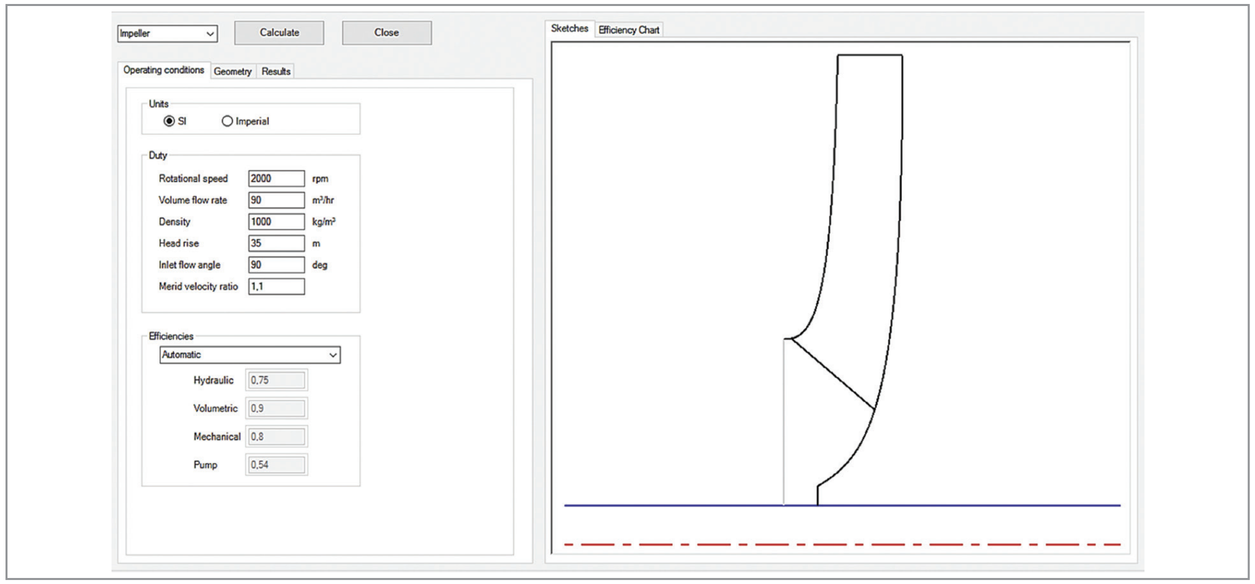


Figure 2. Input design parameters in Vista CPD

The generated geometry is then transferred to ANSYS DesignModeler within the Workbench environment. Figure 3 shows the meridional view of the impeller.

The shape of the impeller (with blades) is complex and can be represented by many parameters. In this case, 29 geometric design parameters are selected to describe the impeller. These parameters can be grouped into:

- Number of blades
- Impeller diameter
- Inlet and outlet diameters
- The location of the leading and trailing edges
- The Bezier points on hub and shroud
- Blade Beta angles on hub and shroud
- Thickness distribution of the hub and shroud layer

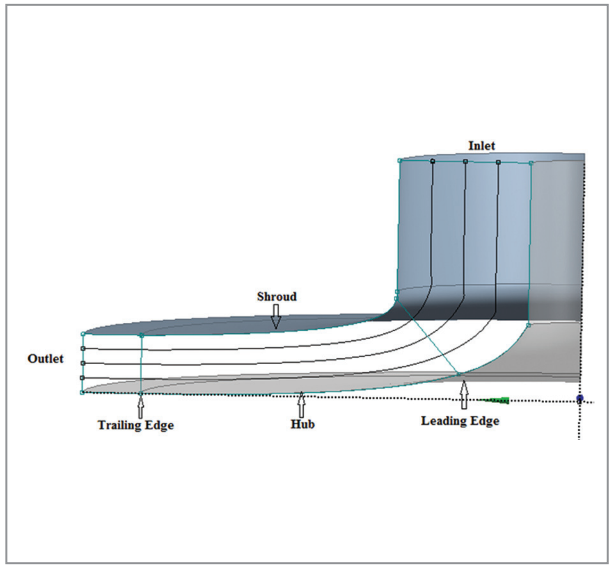


Figure 3. Meridional view of the impeller

2.2. Computational Mesh

The next step following the parametric geometry creation is mesh generation. ANSYS offers a very powerful, turbomachinery specific mesh generation tool, called TurboGrid. It allows high quality hex-mesh around the blades. Figure 4 illustrates the mesh structure used in this work. As can be seen from the figure, the initial mesh was kept coarse (c.a. 200,000 elements for a single blade passage) for a short simulation turnaround time. Once the optimized parameters were obtained, a refined mesh can be used for a more accurate solution.

2.3. Numerical Model

A single blade passage is modeled with rotational periodic boundary conditions, in order to reduce the mesh element count and the simulation turn-around time. The computations are carried out with ANSYS CFX. The flow is taken incompressible, turbulent and at steady-state. The boundary condition is 1 atm at the inlet and 25 kg/s at the outlet with an impeller rotational speed of 2,000 rpm. The working fluid is water at 25° C. The solid boundaries are modeled as no-slip, hydraulically smooth walls. SST k-ω turbulence model is used with automatic wall function. The average value of y^+ is maintained at less than one.

2.4. Optimization Process

Following the CFD simulations, sensitivity analysis and Metamodel Based Optimization (MOP) of the pump impeller are performed. Analyses were performed with the ANSYS optiSLang module included in ANSYS Workbench.

In sensitivity analysis, which constitutes the first step of the optimization process, a metamodel (MOP) is created that can express the objective functions

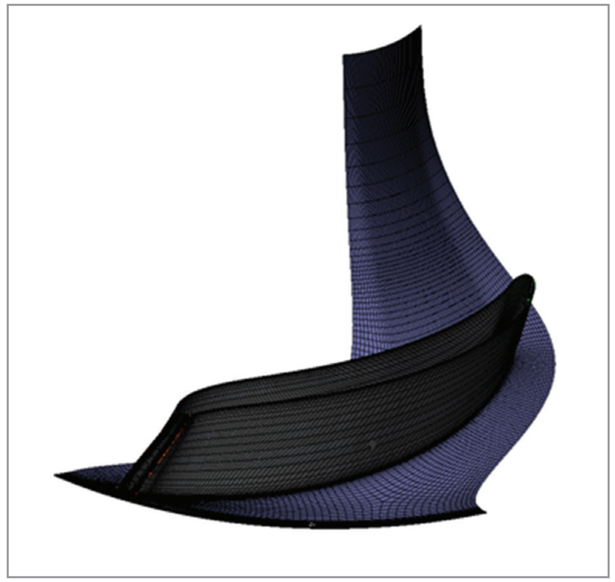


Figure 4. Mesh structure

(output parameters) at the highest quality by using the design points (input parameters). The quality of the metamodel is determined by the so-called Coefficient of Prognosis (CoP) value. Metamodel Based Optimization, the second stage of this study, optimizes MOPs of the objective functions that is obtained in the first stage, allowing optimization to be performed without having to call the CFD solver during optimization. At this point, the optimization time is shortened and the accuracy of the optimized objective functions is increased.

In this study, the objective functions are defined such that the efficiency is maximized and the torque is minimized. Overall 29 design parameters were determined and the constraints were set to be approximately ± 10% of the initially inputted values.

Considering the aforementioned requirements, Sensitivity Analysis were carried out to achieve three different goals:

- Determination of the variation in the input parameters influence on the change of objective functions,
- Filtering of junk input parameters, and
- Creation of MOP.

The Advanced Latin Hypercube sampling method is used to create the Design of Experiment (DoE), wherein sample points are equiprobable, random, and orthogonally distributed in the design space.

The CoP matrix generated as a result of the sensitivity analysis is shown in Figure 5. As can be seen from the figure, the CoP matrix has seven design parameters and two response functions shown at the x and y-axes of the matrix, respectively. The colors of the matrix elements indicate the degree of correlation between the input and output parameters. The grey matrix elements indicate irrelevant input parameters.

In the MOP - based optimization phase, the unimported parameters are filtered by ANSYS optiSLang and the optimization process is continued with fewer design parameters, and hence, the time for optimization process is reduced.

3. RESULTS & DISCUSSIONS

Figure 7 shows the velocity streamlines for the initial and optimized impeller designs. In the initial design, the velocity vectors indicate that the flow does not perfectly follow the blade profile in the impeller passage. Flow exhibits slight irregular behavior. In the optimized design, however, the flow is nicely aligned with the blades.

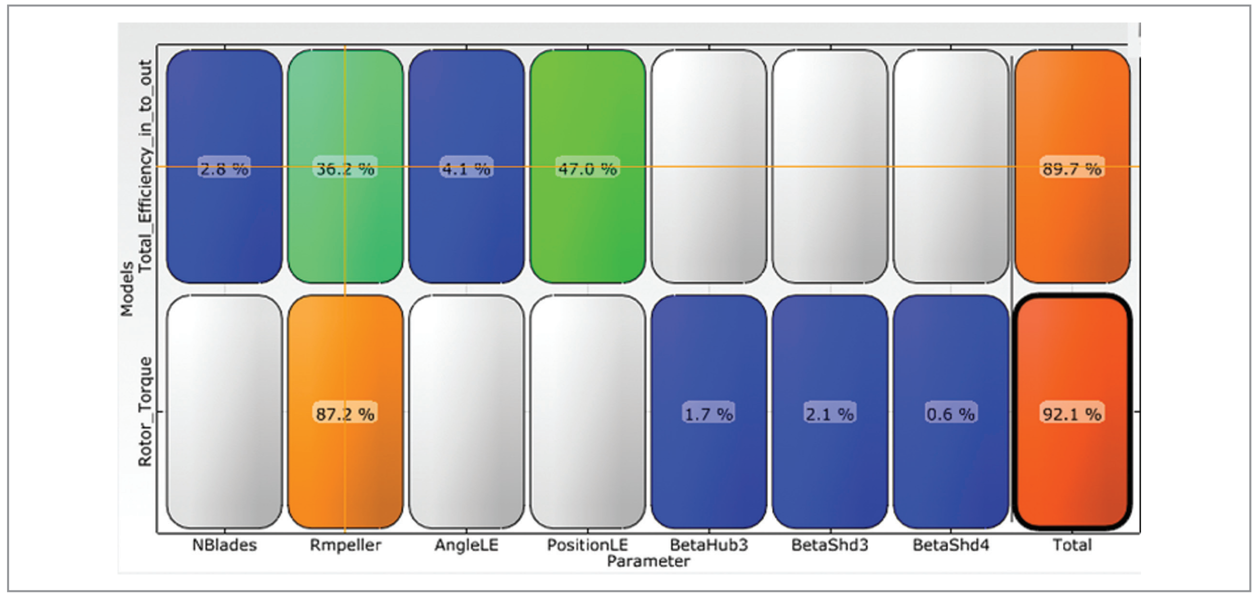


Figure 5. CoP matrix

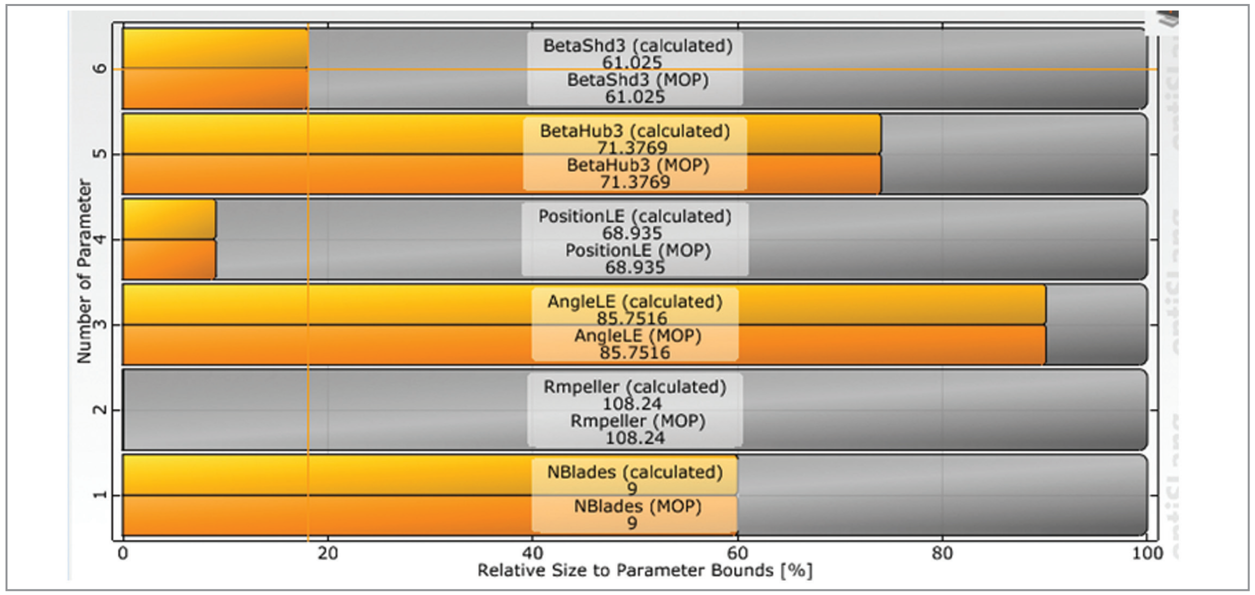


Figure 6. Values of optimized design parameters

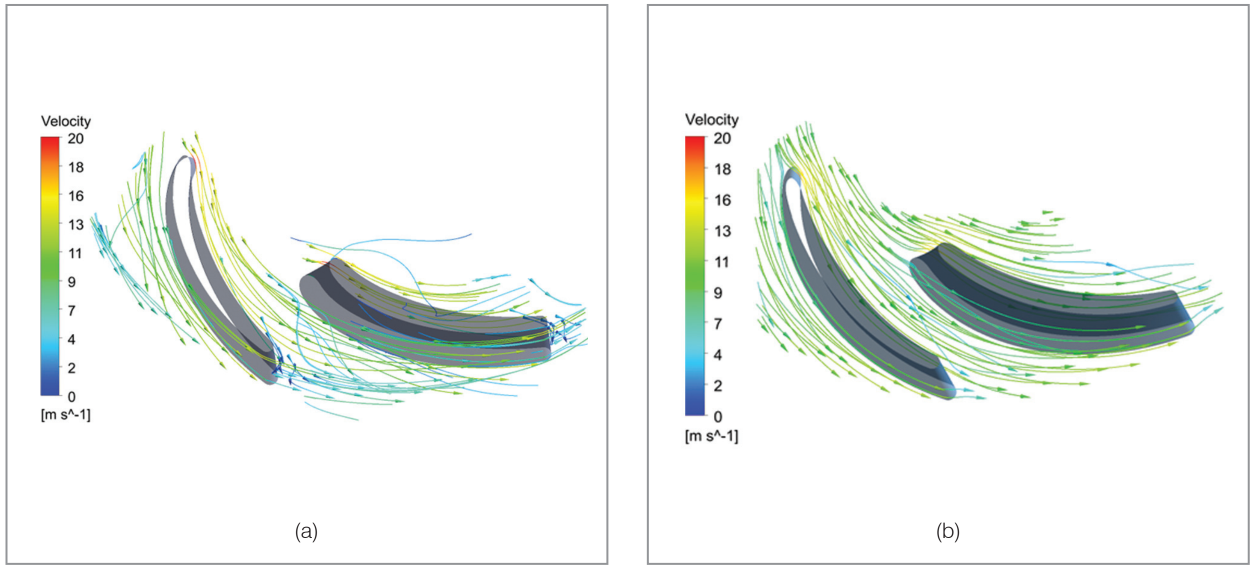


Figure 7. Velocity streamlines for (a) initial design, (b) optimized design

Initial and optimized impeller designs are shown in Figure 8. As can be seen from the figure, the initial design has seven blades whereas the optimized has nine.

Figure 9 shows the meridional velocity for the initial and optimized impeller designs. In the initial design, the velocity vectors indicate that in some regions the flow deviates from the meridional direction. In the optimized design, however, the flow nicely developes in the meridional direction.

Table 1 summarizes the interested pump characteristics for the initial design direct from Vista CPD and the optimized design from optiSLang. From the Table, it is obvious that the optimization process resulted in a decrease in the torque, shaft power and impeller diameter and an increase in efficiency.

Table 1. Analysis results of the initial and optimized impeller designs

	Outer Diameter (mm)	Torque (N.m)	Shaft Power (kW)	Hydraulic Efficiency (%)
Initial Design	243.2	55.1	11.538	75.9
Optimized Design	229.9	45.7	9.558	81.5

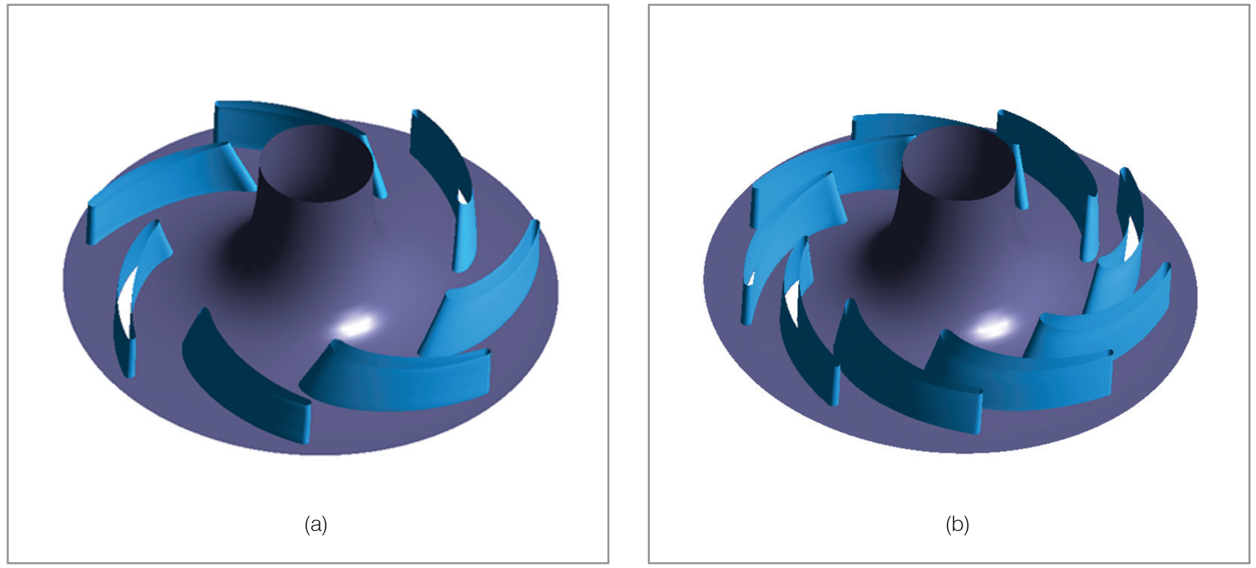


Figure 8. 3-D Models of the impeller (a) initial design (b) optimized design

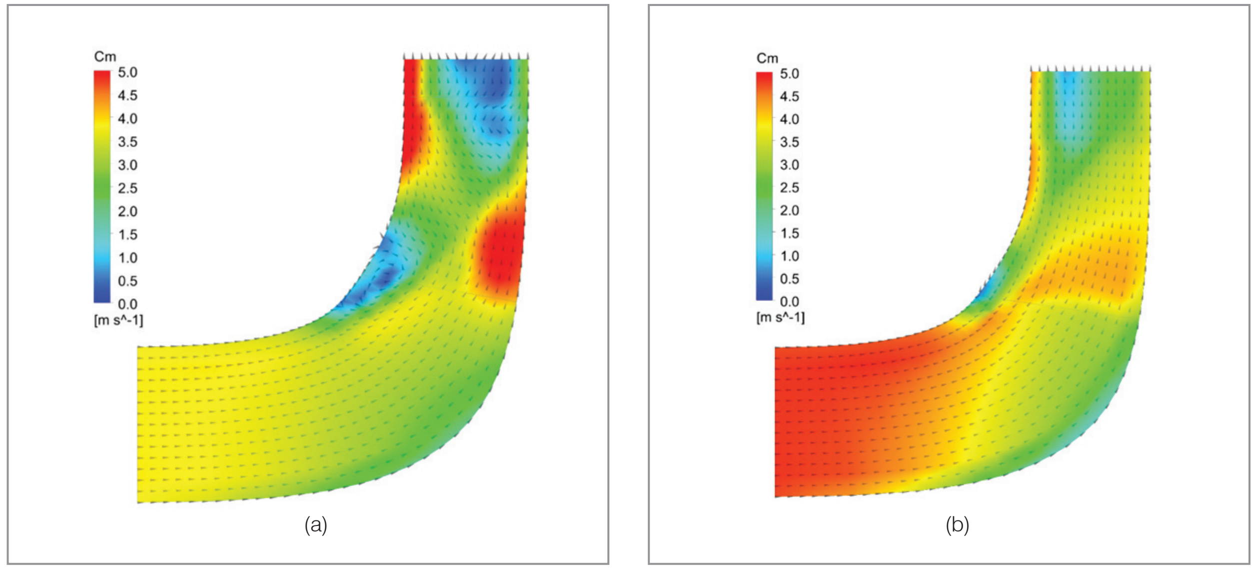


Figure 9. Meridional velocity (a) initial design (b) optimized design

4. CONCLUSIONS

In this study, optimization of a centrifugal pump impeller was carried out to maximize efficiency and minimize torque. The base impeller was designed using Vista CPD and internal design databases, and its performance was evaluated through numerical analysis. Design optimization was then performed to improve the performance of the base impeller by using the optimization tool, called optiSLang. The design parameters and ranges for the impeller were defined. An optimized impeller that satisfies the design specifications was obtained. Finally, the performance of the optimized impeller was verified through detailed numerical analysis.

5. REFERENCES

[1] Gülich J. F., Centrifugal Pumps, Springer Berlin HeidelBerg, 2008

[2] Kreuzfeld G., Müller R., Stübing S., 2014, "Initial Design and Optimization of Turbomachinery using CFturbo and optiSLang", 11th Weimar Optimization and Stochastic Days

[3] Dynardo GmbH, "Introduction to ANSYS optiSLang", optiSLang Training Material