

Initial Design and Optimization of Turbomachinery using CFturbo[®] and optiSLang[®]

Gero Kreuzfeld², Ralph-Peter Müller^{1,2}, Sebastian Stübing^{1*}

¹ CFDnetwork GmbH, München

² CFturbo Software & Engineering GmbH, Dresden

Abstract

Using modern CAE-software tools optimized turbomachinery designs can be generated within a few days or weeks by automated optimization processes. Time and efforts to achieve optimal designs mainly depend on computational resources. And it is very important too to create reasonable initial designs which are close to the desired design points. In this respect a new design process is shown as follows.

The geometry for Turbomachinery components can be described as a parametric design system. So main dimensions, meridional shape of the impeller, blade angles, blade profiles, etc. can be determined as free-form models or other geometrical entities. Additionally fundamental equations combined with proven empirical correlations will help to get good initial design. Furthermore it is essential to have excellent interface available to transfer geometry model from the design system to simulation software package in order to start optimization.

CFturbo[®] is a user-friendly, interactive Turbomachinery design software to create impellers, stators, diffusers and volutes. The software can be used for both: to design new machines or components from scratch as well to re-design machines to get a starting point for modification or optimization. Automated workflows are highly essential for iterative design loops. The user can work manually and intuitively using CFturbo[®] in collaboration to CFD/FEA codes or it can be done automatically when governing such a process by optimization software. Within CFturbo[®] there are interfaces available to all major commercial CAE-simulation systems in order to create seamless workflows for conceptual design, simulation and optimization. As follows a new workflow for automated optimization of an unshrouded compressor impeller is presented using CFturbo[®], ANSYS-Workbench and optiSLang[®].

Keywords: Turbomachinery, Conceptual design, compressor, optimization, automated workflow, CFturbo, ANSYS-CFX, optiSLang

* Kontakt: Dr.-Ing. Sebastian Stübing, CFDnetwork GmbH, Friedrichstraße 20, 80801 München, sebastian.stuebing@cfdnetwork.de

1 Introduction

During the last decades numerous empirical correlations have been developed by examining various turbomachinery measurement data. In the beginning of a new turbomachinery design process only the design point may be known which can be the main operating point of the machine. The engineer has to compute all relevant geometric parameters like main dimensions, number of blades, blade angles and so on to design a Turbomachine that meets all requirements. Applying so-called approximation functions (see **Figure 1**) based on empirical correlation a prediction of optimal or at least acceptable values for required geometrical parameters is possible.

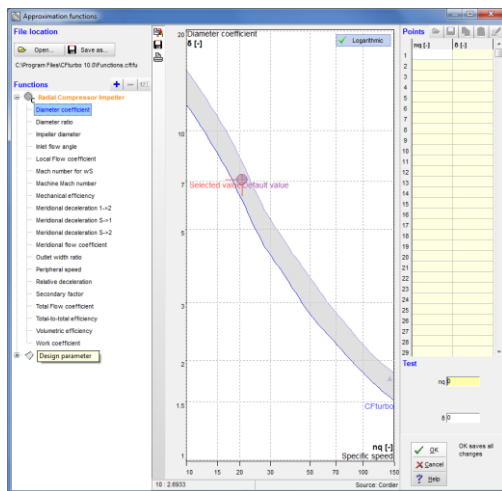


Figure 1 Approximation functions used in CFturbo[®] for initial design (here: diameter coefficient vs. specific speed).

CFturbo[®] includes a multitude of approximation functions based on empirical correlations which should help experienced and unexperienced design engineers to create very reasonable initial turbomachinery designs. Currently over a hundred of different approximation functions are integrated within the software. In addition to these functions it is also possible to replace the standard functions by own empirical correlations which remain proprietary know-how of the users company. So the engineer can rely on own data, he could use design recommendations of CFturbo[®] software or alternatively he could do manual dimensioning without using proposed empirical functions.

Even though many empirical correlations for turbomachinery design parameters are well known and will be available inside CFturbo[®] there are still some unknown relations and parameters which need to be explored or which can be improved significantly. The application of an automated optimization processes can become very useful to find excellent turbomachinery designs. The authors will give a description how to set-up a workflow to optimizing an unshrouded centrifugal compressor impeller. This project should show a general path for initial design, CFD-simulation and optimization using modern CAE-software. Optimization goals, constraints and design parameters have to be defined adequately. The targeted workflow should try different optimization algorithms to reach the best design with a minimal number of simulation loops.

2 Turbomachinery design

CFturbo[®] is an interactive turbomachinery design tool which guides the user through the entire design process of various Turbomachinery components. First the user has to set-up the design point of the Turbomachine. Based on the mass flow rate, required energy transmission, speed and inlet conditions CFturbo[®] software suggests which machine type will fit for the given operating conditions in general. Figure 2 - left hand side - shows the GUI to define global setup. In following steps the user decides about a shrouded or unshrouded impeller, and he has to choose design parameters that will be applied for computing the main dimensions, meridional contours of the impeller, blade angles etc. On the right hand side of Figure 2 a dialogue for designing the meridional shape of the impeller is shown. Additional views for curvature progression, static moment or development of the cross section area should help to design the hub contour and shroud contour. The user may modify all suggestions by his own knowledge at any time. Hence CFturbo[®] is a powerful tool for both, novice and expert design engineers.

To make automated turbomachinery optimization the availability of a parametric geometry model is necessary. CFturbo[®] models meet the essential condition to be used in an automated optimization process since it is fully parametric for all integral values and free-form parameters too. Parametric models will be available for various meshing or simulation tools. In this specific project both meshing and 3-CFD-simulations were conducted within ANSYS Workbench environment using ANSYS-TurboGrid and ANSYS-CFX. Hereafter the simulation workflow and the optimization strategy will be introduced in detail.

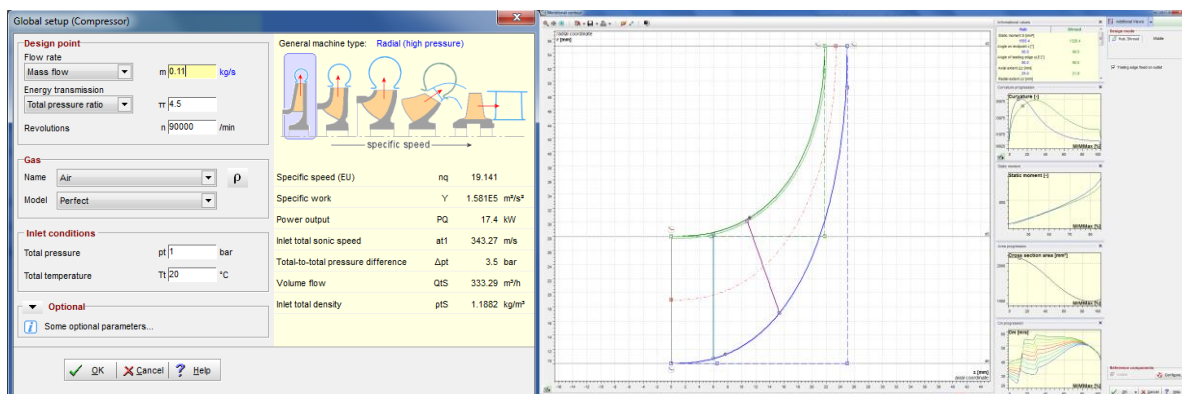


Figure 2: Global setup (left) and meridional view (right) for impeller design process.

3 Numerical model and simulation

First step of the optimization process is to prepare the simulation workflow consisting of geometry preparation, meshing and fluid flow simulation. As shown in **Figure 3** the workflow is partly build within ANSYS-Workbench, using ANSYS-TurboGrid for grid generation and ANSYS-CFX to simulate fluid flow. Currently CFturbo[®] is not yet integrated into the ANSYS-Workbench but this will be available soon. The geometry model is exported from CFturbo[®] using the ANSYS-TurboGrid interface. A session file is written which

contains all information about geometry files (*.curve files). A Python script will be used to update geometry in ANSYS-Workbench. Based on a given mesh resolution ANSYS-TurboGrid is used to generate a computational mesh for the 3D-CFD-simulation automatically.

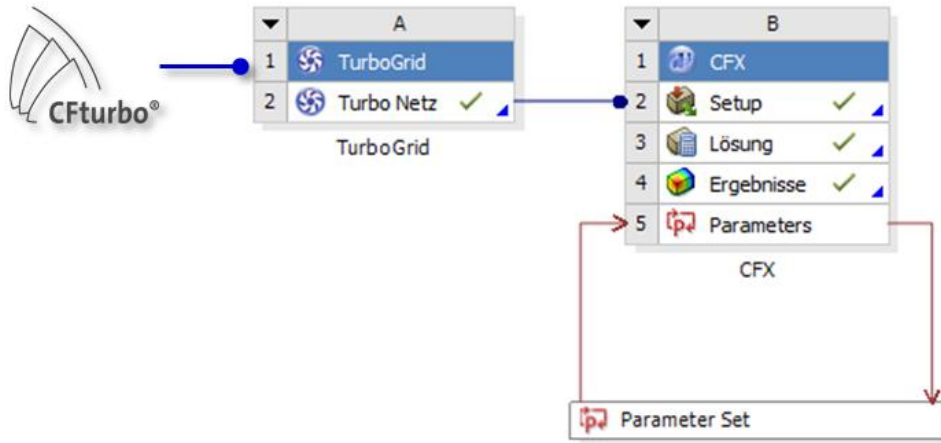


Figure 3 Workflow for design and CFD-simulations using CFturbo[®] and ANSYS-Workbench.

Figure 4 shows the computational mesh of an initial design. Besides impeller the mesh includes also an inlet and outlet domain which have been added for numerical reasons. Therefore an upstream flow channel and a downstream diffuser after the impeller have been created in CFturbo[®]. Of course these stator elements can be meshed by other software tools like the ANSYS-Workbench Mesher, too. The model has included one single passage with periodic boundary conditions and rotor-stator-interfaces (RSI) to the inlet and outlet stator, respectively.

The simulations were performed as steady-state using the frozen-rotor approach. At the inlet the total pressure and temperature were defined as 1 bar and 293 K, while the mass flow was set at the outlet. As only a single segment was calculated, the mass flow had to be corrected by the total amount of passages in a 360° model to meet the overall mass flow rate of 0.11 kg/s for the full stage. For fluid flow simulations air as ideal gas was chosen. The output parameters to be achieved were as follows: pressure ratio Π , isentropic impeller efficiency η and total input power P . These integral values have been determined within ANSYS-CFD-Post.

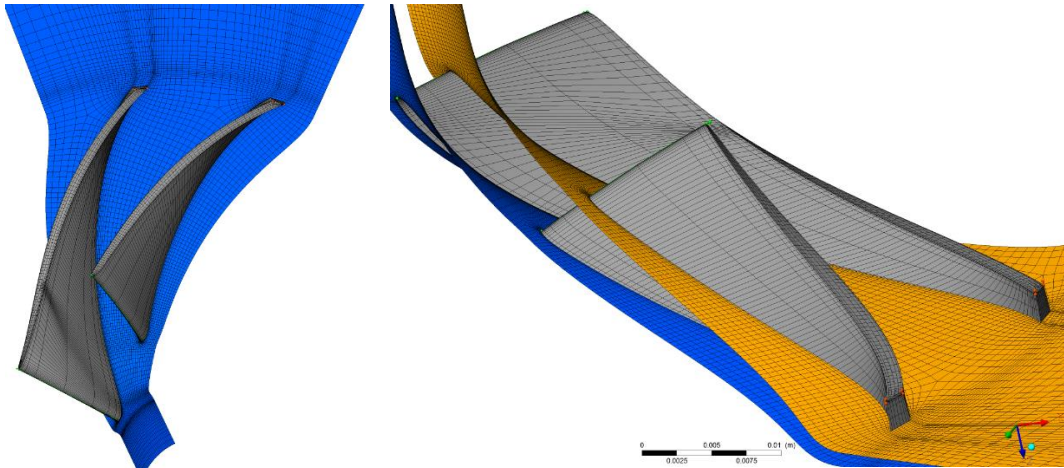


Figure 4 ANSYS-TurboGrid mesh of initial impeller design including inlet and outlet domain

4 Optimization

This project should enhance the process for optimal impeller design under consideration of different constraints. The optimization goal was defined to get an impeller design with a maximum efficiency at design point. The required power P should be lower than 25.5 kW and the pressure ratio Π to be achieved should cover the range $\Pi = 4.5$ to 5.5. Furthermore the outlet angle β_{B2} should be in the range between 20° and 90° to ensure backward curved blades due to efficiency reasons. So more than 20 design parameters have been defined which could be varied by the optimizer software optiSLang[®] (s. **Table 1**).

Table 1: Geometry parameters as used in optimization workflow.

Main dimensions	Blade properties
Suction diameter	Number of blades
Impeller diameter	Incidence factor
Outlet width	β_{B2} on hub and shroud
	Trailing edge main blade
	Trailing edge splitter blade
	Relative position of splitter blade
	Wrap angle for main and splitter blade
	2 Bezier-points for main and splitter blade

First a sensitivity analysis was started to detect and to separate the design parameters which have no or only a minor influence on the impeller performance and efficiency. So optiSLang[®] should be enabled to search for the efficiency optimum using a significantly reduced set of design parameters which results in much less simulation runs. **Figure 5** shows a sketch of the applied workflow within optiSLang[®] of Dynardo GmbH

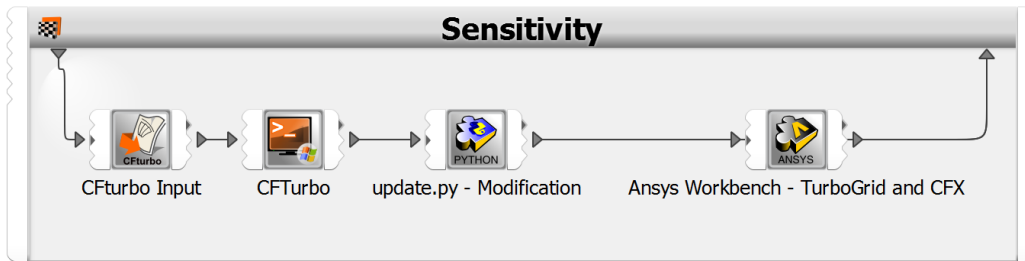


Figure 5: Workflow within optiSLang®.

The sensitivity analysis applied an ALHS (Advanced Latin Hypercube Sampling) algorithm, which uses a random sampling of design points to be examined. During sensitivity analysis more than 50% of the generated designs failed, either because no geometry could be created by CFTurbo® or due to meshing problems. This number is not unusual in new developed CAE-optimization projects. A more appropriate choice of geometric parameter combinations in the beginning would lead to less failed designs. In general the optimizer has to be given a maximum degree of freedom regarding the parameter field. In this special case some parameters had to be restricted whilst dependencies between the parameters were defined. So the amount of failed designs could be reduced significantly.



Figure 6 Impeller geometries (left: initial, right: optimized design) created by CFTurbo®.

With the best design out of sensitivity analysis an increase of impeller efficiency from initially 78% to 78.5% was achieved. It shows that random sampling is not helpful in this special case. In consequence the design space for the optimization was adjusted and another optimizing algorithm was chosen. By using ARSM (Adaptive Response Surface Method) algorithm in a target area close to initial design the impeller efficiency was increased to 84.5% with about 100 simulations only. **Figure 6** shows impeller geometries of the initial design on the left hand side and the improved design by the optimizer on the right hand side. The number of blades of the optimized design ($n=22$) is higher than of the initial one ($n=16$). And especially the length and the splitter blade position would probably not have been chosen in a manual design and modification process. **Figure 7** is showing flow patterns of air flow in blade-to-blade-view comparing base design and optimized design. The optimized geometry shows less recirculation on the blades. By reducing these losses the overall impeller efficiency could be increased.

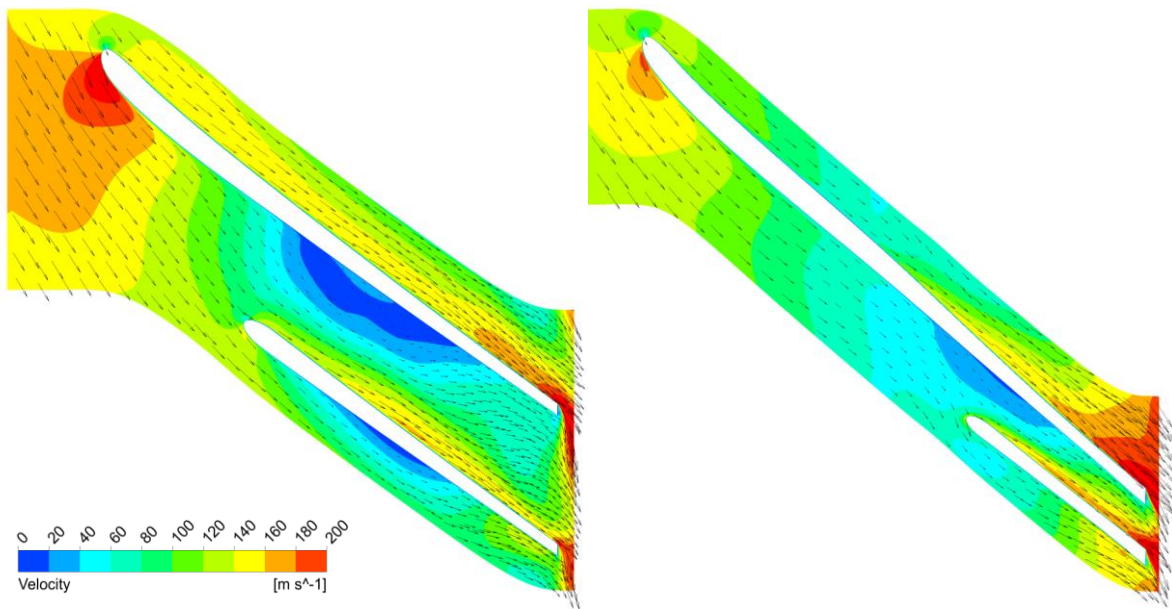


Figure 7: Fluid flow velocity magnitude through the blade passage. On the left the initial design is shown and on the right hand side the optimized design.

Using an automated design, simulation and optimization process can provide better results in a shorter time than it could be achieved by manual-intuitive methods. By understanding of existing correlations or by determining new relations between different design parameters on one side and turbomachinery performance and efficiency on the other automated optimization workflows will make a considerable contribution. New and - so far - unusual approaches in turbomachinery design can be taken to benefit Turbomachinery researchers, engineers, manufacturers, and end-users as well.

5 Conclusion

This work introduced a new, stable and reliable workflow for turbomachinery design optimization using CFturbo[®], ANSYS-Workbench and optiSLang[®] by Dynardo. It was a learning process. State of the art turbomachinery design principles which are implemented in CFturbo[®] will allow to create good initial designs of impellers and other Turbomachinery components. By choosing appropriate optimization parameters and dependencies the number of failed or useless designs can be reduced significantly. optiSLang[®] can be applied to use algorithms finding local optima which will reduce the number of simulations additionally. Finally a remarkable improvement in impeller efficiency has been reached with a quite low number of simulations. Future developments of CFturbo[®] software will establish optimization workflows including additional turbomachinery components like stators, diffusers or volutes.