AUTOMATED OPTIMIZATION OF CENTRIFUGAL COMPRESSORS "ECKARDT ROTOR O"

Aly M. Elzahaby
Professor,
Mechanical power Engineering
Department
Tanta, Egypt

Mohamed K. Khalil
Lecturer,
MTC A/C Mechanical
Engineering Department
Cairo, Egypt

Mohammed E. AlMoghazy
Teaching Assistant
Mechanical Engineering
Department
Kafr ElSheikh, Egypt

ABSTRACT

Centrifugal compressors play an important role in many industries. Improving the efficiency of centrifugal compressors and extending their range has been an important subject for both engineers and researchers working in the turbomachinery field. This paper discusses the optimization of Eckardt O impeller through changing its blade angles distribution to increase its efficiency. the optimization process is performed using an automated procedure performed within ANSYS workbench. The geometry is parameterized using ANSYS design modeler, the mesh is generated using ANSYS Turbogrid and steady flow CFD results are obtained using ANSYS CFX. Optimization by genetic algorithm is done using a surrogate model generated through a sample of designs selected through Design of experiments "DOE" sampling. The performance of the optimized and the original designs are compared both qualitatively and quantitatively, the optimized design efficiency successfully increased from 87.994 % to 88.481% based on CFD results.

NOMENCLATURE

p_0	Total pressure, Pa
T_0	Total temperature, K
R	Radius, m
PR	Pressure ratio

Greek symbols

η	Efficiency
β	Relative blade angle
Θ	Corrected total temperature
δ	Corrected total pressure
$\dot{m}_{cor}$	Corrected mass flow rate, kg/s
n	Rotational speed, rpm

Subscript

tt	Total to total

Abbreviations

DOE	Design of Experiments

INTRODUCTION

Centrifugal compressors are characterized by their high-pressure ratio per stage, robustness and stability over a wide operating range [1]. These characteristics make them ideal for many important applications including sophisticated industrial, aerospace and automotive industries. Designers of centrifugal compressors use the latest tools and methodologies to design aerodynamically efficient compressors that meet the structural, manufacturability and costs constraints imposed by the specific application for which they are designed. Designing a centrifugal compressor rotor starts from calculating its main dimensions, this is done utilizing simple 1D or 2D methods [2]. The next stage is defining the 3D geometry of the compressor blades. The 3D blade geometry is specified through the definition of the hub, shroud, blade angle distribution and thickness distribution curves. These curves are generally defined as Bezier or spline curves for which the coordinates of the defining points are parameterized [3],[4]. There are no available analytical methods to specify the blade defining curves to give optimal performance. It is rather an iterative process where an optimization workflow is used [2]. The optimizer varies the parameters controlling the blade defining curves. The aerodynamic performance is calculated using CFD. Finally, the optimizer introduces the best candidate designs according to the predetermined objective function and constraints.

For optimization, surrogates or metamodels are used to approximate computationally expensive functions to provide computationally inexpensive reasonable predictions of the real functions. Standard polynomial response surfaces, Kringing model and artificial neural networks are popular in the turbomachinery design discipline. The design of experiments "DOE" techniques are used to introduce proper samples to construct an approximate metamodel that is as accurate as possible and covers the required region of the design space with the least design points to save the computational resources[5], [6].

In this paper optimization of the Eckardt O compressor is introduced. The blade angle distributions curves of the Eckardt O are redesigned to get optimum aerodynamic performance at the operating point of the original Eckardt O compressor. The workflow is performed using ANSYS software package and optimization is performed using surrogate model and Design of experiments methods.

AUTOMATED OPTIMIZATION WORK FLOW

The automated optimization workflow scheme, Figure 1, includes the following modules:

- **Geometry module**: this module contains the parametric CAD geometry to be controlled through the accessible geometrical parameters.

- **TurboGrid module**: uses the geometry in the geometry module as an input and generates a suitable mesh according to a set of user-defined settings.

- **CFX module:** This module calculates the 3D flow field and calculates specific centrifugal compressor performance parameters according to a set of predefined equations.

- **Response surface optimization module:** Optimizes the compressor geometry to get the best performance according to a user-defined objective function and without violating a set of pre-defined constraints. It uses the response surface model and design of experiments for sampling.

The optimization process works in the following order: the optimization module selects a sample of compressor geometries which cover the entire design space, this set is defined using a suitable design of experiments techniques. The geometry is used to generate a suitable mesh, then the performance of the design is calculated, and the output performance parameters are stored. This process continues till the performance of each of the designs in the initial sample is calculated. The stored data is used by the optimization module to generate a surrogate model (also called a response surface) with the compressor performance parameters as outputs (responses) and geometrical variables as inputs.
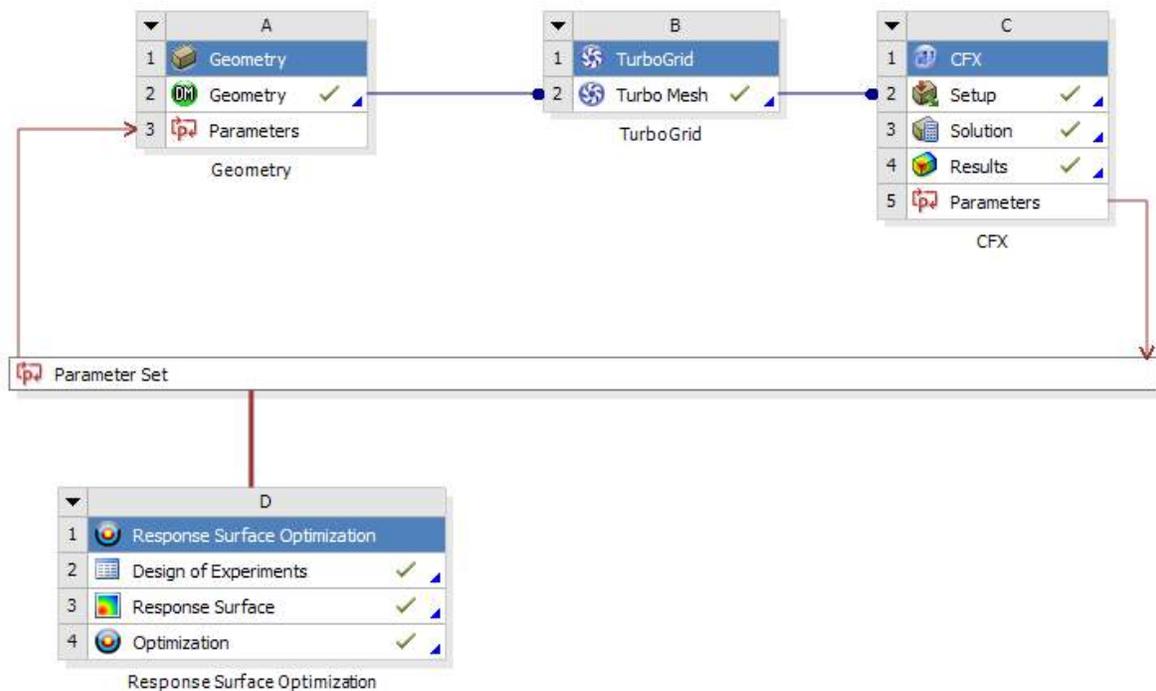

Figure 1: Automated optimization workflow schematic in ANSYS workbench

Using an automated optimization workflow saves time and effort, but the following requirements should be fulfilled so that the automated workflow can work:

- Preparation of a robust parametric CAD model of compressor.
- Setting up a mesh configuration that achieves a good balance between accuracy, computational time and robustness.
- Proper specification of the CFX setup, solution parameters and defining the equations for the calculated performance parameters.
- Validation of the CFX setup and solver settings through using the Eckard O compressor geometry and comparing the calculated parameters to the experimental results.

GEOMETRICAL MODEL

Two CAD models are required for the analysis, the first one is the Eckardt O compressor 3D model, it is required for the CFD setup validation step, the second model is the parametric model used in the automated optimization workflow.

Rotor of the original Eckardt O compressor

The Eckardt Rotor O geometry was modeled using siemens NX CAD package, the rotor hub and shroud curves are circular arcs, blade elements are radial, and the blade camber lines are elliptical curves in cylindrical sections. Figure 2 shows the main dimensions of the meridional contour of the Eckardt O rotor, the elliptical blade camber curves in cylindrical sections are defined using the following equation:

$$\frac{(Y-a)^2}{a^2} + \frac{X^2}{b^2} = 1$$

(1)

Where: a = 4.7693R mm, b= 220.579 mm

No data was found in the literature regarding the blade thickness of the Eckardt O rotor, so it was assumed following the design recommendations used at the time the original compressor was designed, Blade thickness was assumed constant for stream-wise sections and increasing linearly from the shroud to the hub along the blade span, the blade thickness was assumed 1.25 mm at the shroud and 2.5mm at the hub. Figure 3 shows the created CAD model of the compressor.
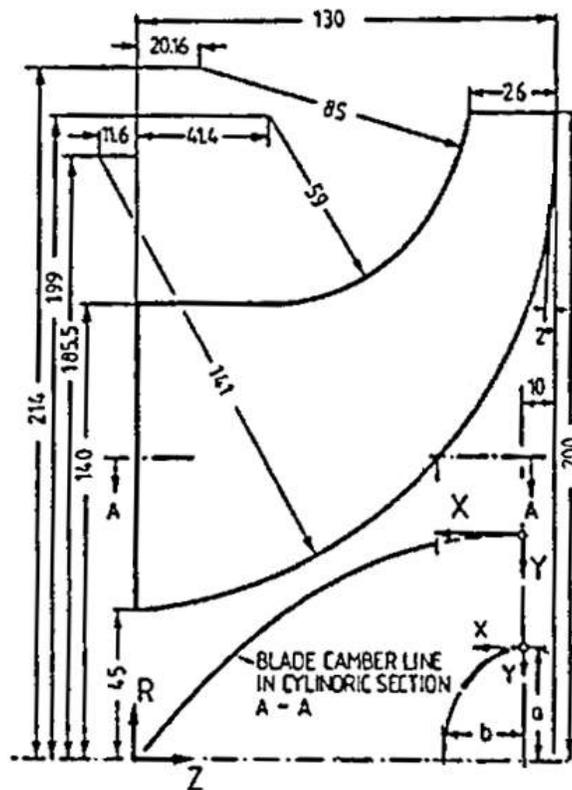

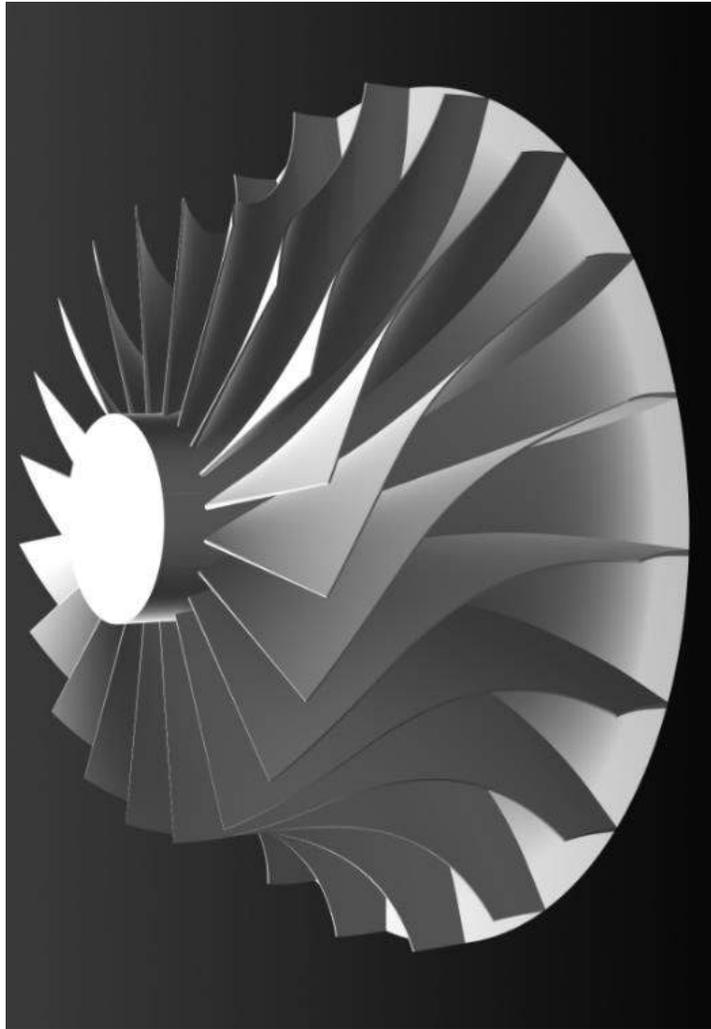*Figure 2: Meridional contour and mean camber line* [7]

Figure 3: The developed CAD model of Eckardt O rotor

Rotor of the parametric CAD model

The parametric CAD model used in the optimization workflow is defined in the ANSYS design modeler module. It shares the same meridional contour with the original Eckardt O rotor. For the blade, Radial element surface construction is used, this makes the definition of the blade camber line at only one cylindrical section at any radius from the rotor axis sufficient to define the whole blade camber surface. In ANSYS blade modeler the blade camber lines can be defined at stream-wise sections by defining their blade angles values along the meridional length [4]. For the developed parametric CAD model, A Bezier curve with 6 control points was used to define the blade stream-wise camber line at the shroud section. The range through which the coordinates of each of the control points could be varied is carefully selected to avoid the generation of distorted blade geometries. three of the control point are kept fixed and the meridional coordinates of the other three points were also kept fixed, leaving only three parameters controlling blade angles as variables to be controlled by the optimizer during the automated optimization workflow. Figure 4 shows the blade angles distribution at the shroud stream-wise section, the control points designated by "+"

kept fixed and only the angle values for the control points designated by "o" are used as parameters to be controlled by the optimizer, these parameters are named beta1, beta2, beta3 and the range of their values is specified in Table 1.

Table 1 upper and lower bounds of geometrical parameters

Variable	Lower bound	Upper bound
beta1	-30º	-15º
beta2	-30º	-5º
beta3	-5º	5º

The thickness distribution applied was assumed in the same way previously used to create the Eckardt O CAD model.
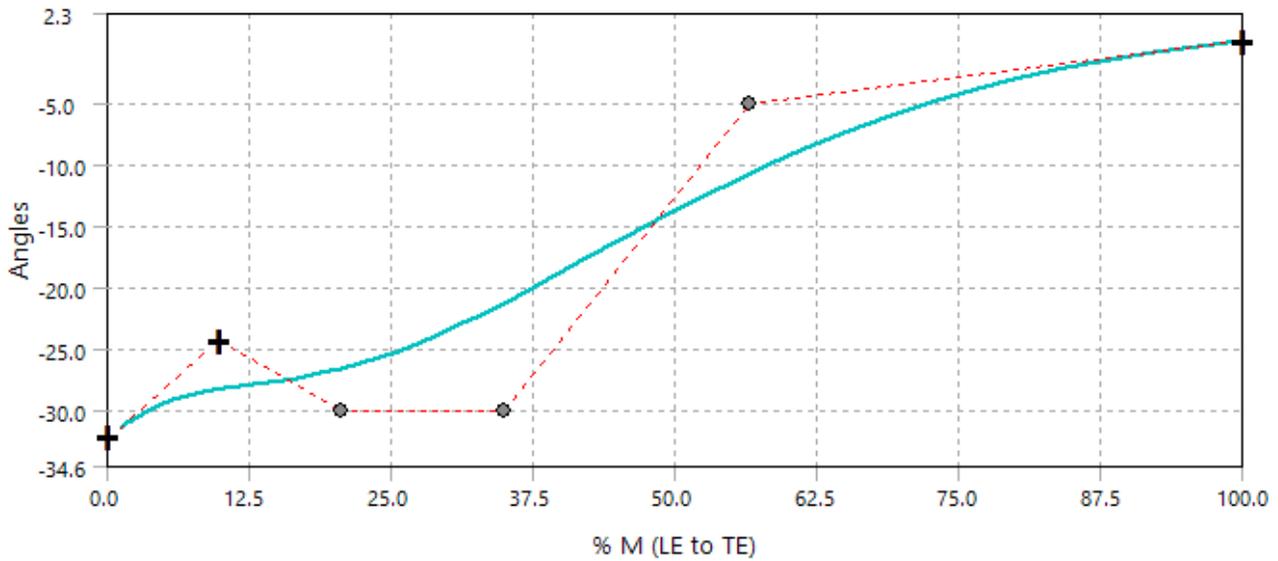

Figure 4: Blade angle distribution at the shroud stream-wise section

Diffuser Geometry for the CFD verification CAD model

The original Eckardt O diffuser is vaneless with constant flow area up to a radius equal to twice the impeller tip radius (R/R2 = 2), followed by a bend to turn the flow into an annular settling chamber as shown in Figure 5, the experimental results reported are based on inlet conditions at station 0 and outlet conditions at station 4 (R/R2 = 1.69) [8], thus, the vaneless diffuser used in the CAD setup verification was modeled only up R/R2 = 1.69 such that the CFD results could be validated against the published experimental results. The same diffuser geometry was also used with the parametric rotor model used in the automated optimization work flow such that the results obtained could be directly compared to the experimental results achieved by the original Eckardt rotor O. Figure 6 shows the meridional contour of the flow path in ANSYS design modeler.
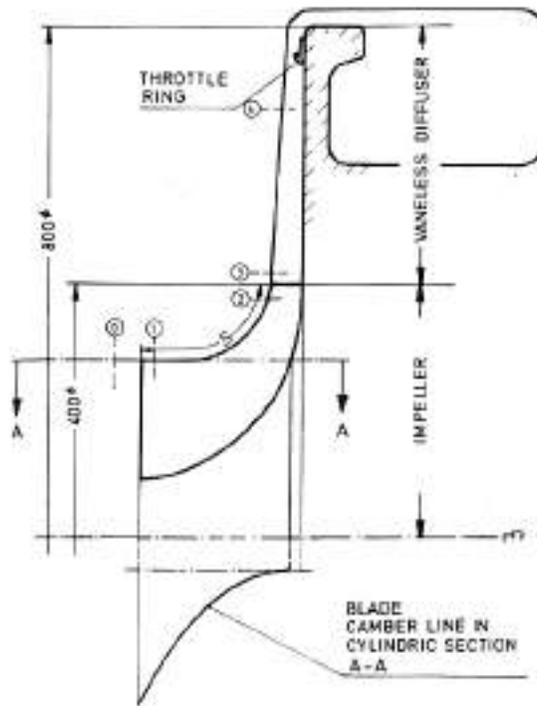
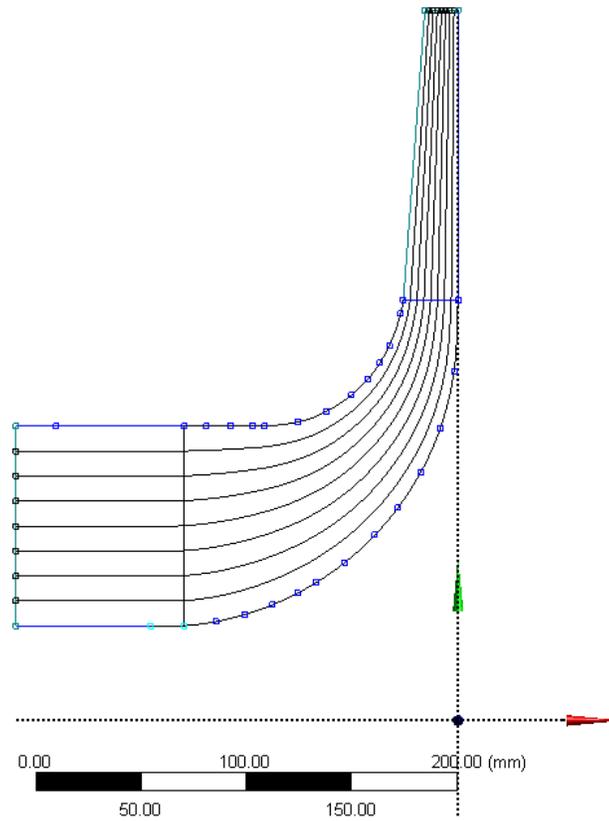*Figure 5: Meridional contour of the compressor stage* [9]

Figure 6: Meridional contour of the flow path in ANSYS design modeler

CFD MODEL SETUP & VALIDATION

The selected computational tool for CFD calculation is ANSYS CFX®. This commercial simulation software package is selected for its proven accuracy in turbomachinery applications including centrifugal compressors, also it can be used integrally with geometry, meshing and optimization modules within the ANSYS workbench environment to create a stable integrated and automated optimization workflow.

Frame of reference

A single frame of reference can be used when calculating the compressor stage, that is, a frame of reference that is fixed with respect to the rotor, this makes the flow steady with respect to this frame of reference which greatly simplifies the analysis, despite the fact that the vaneless diffuser is rotating with respect to this single frame of reference, using the single frame of reference is still possible because the vaneless diffuser consists of only two surfaces of revolution around the impeller axis of rotation [10], using a rotating frame of reference for the rotor and a stationary frame of reference for the diffuser is also possible by adding a suitable interface, but it adds unnecessary complexity to the analysis and yields almost no differences in the results.

Boundary conditions

The flow simulation is performed using a single flow passage between two adjacent blades by using rotational periodicity to calculate the performance of the whole compressor, this reduces the computational time significantly and gives accurate results. Figure 7 shows the computational domain boundaries definitions.

Inlet Boundary conditions:

- Total pressure at inlet (P_0) = 101325 Pa
- Total Inlet Temperature (T_0) = 288.15 K

Outlet Boundary conditions:

- Mass Flow rate = 5.31 kg/s

Wall Boundary condition:

- Blade surfaces.
- Inlet Hub and impeller hub surfaces.
- Diffuser hub surface (with wall velocity: counter rotating).
- Shroud (with wall velocity: counter rotating).

Periodicity Boundary conditions:

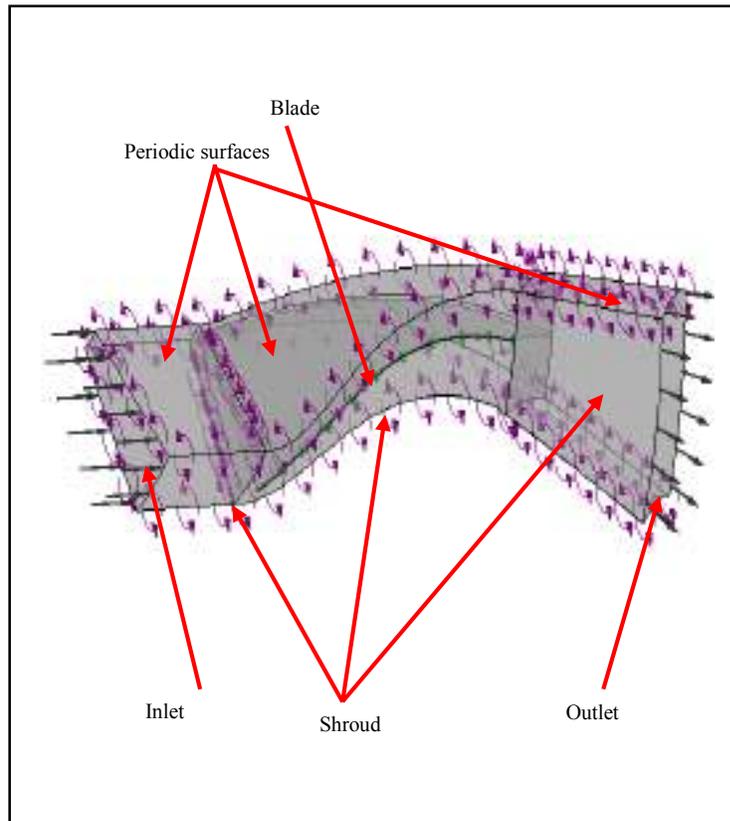- Between the periodic sides 1 and 2 of the hub, passage and shroud.

Figure 7 Boundaries definition

Turbulence model selection

For an automatic optimization workflow, many designs are evaluated using a computationally expensive CFD solver, the simulation time and robustness are both important factors. Accuracy of the CFD solver is also critical to obtain a design that performs as expected. Investigators compared between different turbulence models for centrifugal compressors applications [11] SST model proved to offer a good balance between robustness, speed and accuracy. It is also recommended by ANSYS for turbomachinery applications [12], so it was chosen for this analysis.

Mesh generation.

Mesh generation is carried out using ANSYS TurboGrid, whereas structured type mesh is used. ANSYS TurboGrid has the capability of producing high quality meshes with the least number of elements, thus helps in obtaining fast and accurate results.

Method: Global size factor, this was varied from 1.1 to 1.3 during the grid independence study, the final mesh has a global size factor of 1.1.

Topology: The mesh topology is defined using the ATM optimized option in ANSYS TurboGrid and the selected method is the single round cut-off refined, Figure 8

Boundary Layer: Different methods for near wall element size specification during the grid independence study, for the final mesh an absolute First element offset of 0.5 mm was applied with target maximum expansion rate of 1.3.

Passage data: the method is "Proportional" with Factor =1.

Inlet domain: mesh type is H-Grid, defined by a maximum expansion rate of 1.2, the total node count of the final mesh is 800000 and it proved to be valid through the grid independence test.
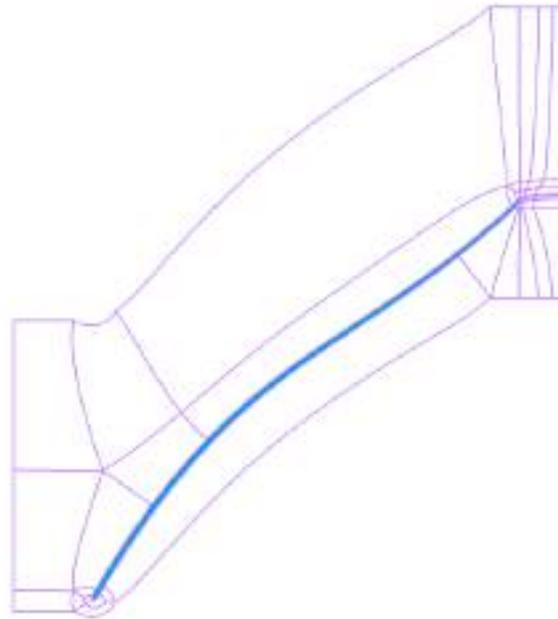

Figure 8 Turbogrid Mesh Topology (Single Round Cut-off refined)

Grid independence study

 Grid independence study is performed at the performance point m =5.31 kg/s at 14000 rpm for the Eckardt O rotor, Table 2. The experimental values of the total to total pressure ratio and isentropic efficiency are 2.094 and 88% respectively. Therefore, the mesh with 8E05 nodes is selected as the final mesh which gives relative errors of -0.468 %for pressure ratio prediction and -0.0068% for efficiency compared to the experimental values.

Table 2 Grid independence study

Passage mesh elements	Total temperature ratio	Total pressure ratio	η_{tt}	Compressor mass flow rate
400000	1.2672	2.0816	87.849	5.31
800000	1.2672	2.0842	87.994	5.31
1100000	1.2665	2.0825	88.13	5.31
1800000	1.2669	2.0849	88.16	5.31

The produced mesh offers Non-dimensional wall distance values, "Y+" suitable for good accuracy with the SST model, Figure 9 shows the Y+ contours on the hub and the blades, the y+ value doesn't exceed 5 almost over all the surfaces
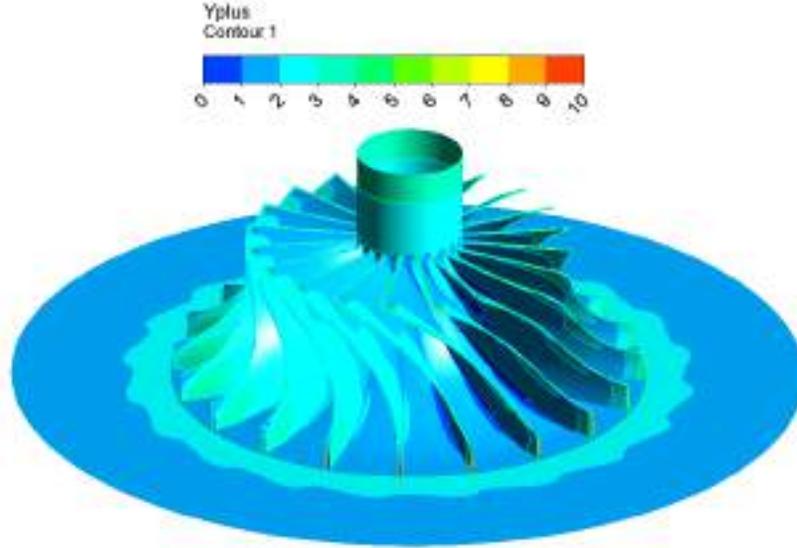
Figure 9 Y plus contours on the rotor and diffuser hub

CFX-pre-setup

High resolution option is selected for advection scheme and turbulence numerics, the maximum number of iterations is set to 500 (the solver terminates even if the convergence criteria is still not met). Automatic time scale factor is used. Timescale factor is set to 5.

The convergence is achieved if the RMS residuals reach 1E-07, a conservation target is set to 0.01 and interrupt control is activated, it defines a better criterion for convergence compared to the RMS residuals. It terminates the solution earlier if all the following conditions are met:

- Current time step>20
- Standard deviation over a moving interval of 40 iterations of isentropic efficiency >0.05.
- Standard deviation over a moving interval of 40 iterations of outlet total pressure <1 Pa
- Standard deviation over a moving interval of 40 iterations of stage mass flow rate < 0.05 kg/s

Calculated parameters

Total pressure ratio: $PR_{tt_4} = \frac{P_{t_4}}{P_{t_1}}$

Corrected mass flow rate: $\dot{m}_{cor} = \dot{m}\sqrt{\theta_0}/\delta_0$

Corrected total temperature: $\theta_0 = \frac{T_{t_0}}{288.15\ K}$

Corrected total pressure: $\delta_0 = \frac{P_{t_0}}{1\ atm}$

Total to total isentropic efficiency: $\eta_{tt\,04} = \frac{\left(P_{t_4}/P_{t_1}\right)^{\frac{k-1}{k}}-1}{T_{t_4}-T_{t_1}}$

57

Computational tool validation

The validation process starts with the comparison between CFD results and the experimental results at n = 14000 rpm, reported by D. Eckardt, the performance parameters of the compressor at the three test points are summarized in Table 3.

Table 3 Summary of Eckardt's O performance at the CFD1, CFD2 & CFD3 Test points

Test Point	$n/\sqrt{\Theta_0}$ [rpm]	$\dot{m}\sqrt{\Theta_0}/\delta_0$ [kg/s]	PR_{tt04} [--]			η_{tt04} [%]		
			Measured	CFD	Relative Error	Measured	CFD	Relative Error
1	14000	6.09	2.061	2.0605	-0.024 %	86.8	87.399	0.69 %
2	14000	5.31	2.094	2.0842	-0.468 %	88.0	87.994	-0.0068%
3	14000	4.53	2.086	2.0608	-1.208 %	86.5	86.531	0.035%

Comparison between measured and CFD impeller tip conditions

Table 4 shows the comparison between the CFD and Experimental impeller tip conditions reported by Japikse & Baines [1],

Table 4 Experimental and calculated impeller tip conditions

	Measured	CFD	Deviation
T_{t_2}	363.5 K	364.7 K	0.330%
p_{t_2}	220.5 kPa	216.314 kPa	-1.898%
p_2	144.7 kPa	143.422 kPa	-0.883%

The results obtained show a good agreement between experimental and calculated CFD results, thus the illustrated mesh settings are applied to the CFD calculations within the automated optimization workflow

OPTIMIZATION MODULE SETUP

Design of experiments (DOE)

The first step is the determination of the upper and lower bounds of the input parameters, the upper and lower bounds are specified in Table 1. The selected design of experiments type is the "Latin Hypercube Sampling Design", whereas the selected samples type is "CCD ". The number of design points generated using this methodology is 15 design points. The output parameters calculated and stored for each design point are the mass flow rate, total to total isentropic efficiency and total to total pressure ratio.

Response surface

Kringing auto refinement is used for response surfaces generation. This method uses the CFD results of the DOE samples for metamodel construction, then checks the accuracy of the response surface through comparing the CFD calculated results of a new, automatically selected performance point to the predicted results from the current metamodel, if the results give an error exceeding the user defined maximum relative error, more automatically selected, CFD calculated performance points are added to construct a more accurate metamodel, this process continues till the response surface accuracy reaches the user defined target. in this analysis the maximum relative error specified is 5%.

Optimization objectives & constraints

The objective of the optimization process is to maximize the total to total isentropic efficiency of the compressor while keeping the total to total pressure ratio constrained between 2 and 2.2. Figure 10 shows the objectives and constrains set in the design explorer component.

Figure 10 Optimization objectives and constraints

Optimization Algorithm and results

The Multi-Objective Genetic Algorithm (MOGA) is used, it is a variant of the NSGA-II algorithm, it aims at finding the global optimum and supports multiple objectives and constraints.

The optimization component is configured to generate initial 500 samples using 150 samples per iteration to find three candidates that deliver the best performance, the performance predicted through the response surface is verified via CFD calculations. The obtained optimum design has the following geometrical variables values, Table 5.

Table 5: Optimum design geometrical variables values

Variable	Value (Degree)
beta1	-15º
beta2	-5º
beta3	0.12º

RESULTS & DISCUSSION

The selected optimized design has the highest efficiency in the design space, compared to the performance of the original Eckardt rotor under the same operating conditions (N =14000 rpm & $\dot{m}$ = 5.31 kg/s). The efficiency has increased from 87.994 % to 88.481%, that is, about 0.487 %. Changing the blade angles distribution improved the blade loading leading to less losses and improved efficiency.

Figure 11 and Figure 12 show a comparison between the blade to blade Mach number contours of the original and the optimized compressors at 60 % span. It is obvious that changing the blade angle distribution changes the blade shape such that the Mach number inside the passage generally decreases leading to a reduction in losses and increase in efficiency. The Mach number reduction is more obvious at 90% span. Comparing Figure 13 and Figure 14 a significant reduction in the Mach number in the inducer section is seen, leading to efficiency improvement.

it can be noticed that the wake region seems to be larger in the optimized impeller, but its severity is less compared to the wake in the original impeller.
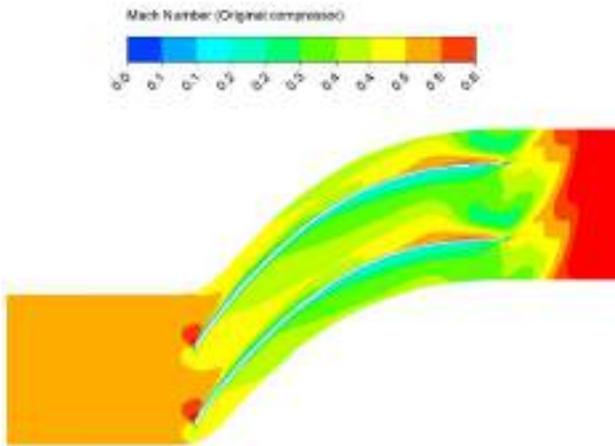
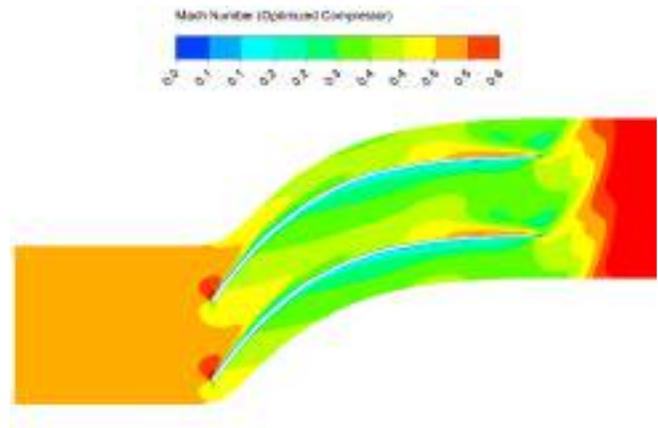*Figure 11: Original compressor blade to blade Mach No. Contours at 60% span*

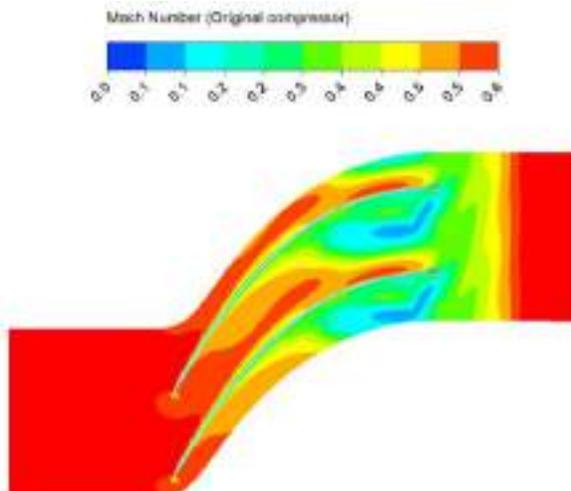*Figure 12 Optimized compressor Blade to blade Mach No. Contours at 60% span*

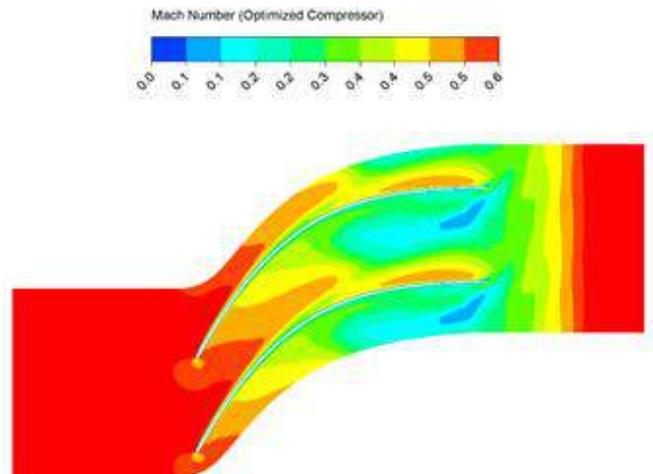*Figure 13: Original compressor Blade to blade Mach No. Contours at 90% span*

Figure 14: Optimized compressor Blade to blade Mach No. Contours at 90% span

CONCLUSION

The optimization of the rotor of the Eckardt O compressor is performed through variation of the blade angle distribution while keeping the blade angles at the impeller inlet and outlet unchanged. An automated optimization is performed based on a surrogate model exploiting DOE for sampling and using CFX as the fluid flow solver. The design point performance for the original and optimized compressors are evaluated and compared both quantitatively and qualitatively, where 0.487% of efficiency improvement is reached. It can be shown that varying the blade angles distribution modifies the blade loading such that a sound improvement in the Mach number distribution is achieved leading to a mild improvement in the compressor total to total efficiency for the same pressure ratio and mass flow rate.

REFERENCES

[1]   D. Japikse and N. C. Baines, *Introduction to turbomachinery*. Concepts ETI, 1994.

[2]   D. Japikse, *Centrifugal compressor design and performance*. Concepts ETI, 1996.

[3]   R. H. Aungier, *Centrifugal compressors : a strategy for aerodynamic design and analysis*. ASME Press, 2000.

[4]   ANSYS, "ANSYS TurboSystem User Guide," vol. 15317, no. October. pp. 724–746, 2012.

[5]   Z. Li and X. Zheng, "Review of design optimization methods for turbomachinery aerodynamics," *Prog. Aerosp. Sci.*, no. May, p. , 2017.

[6]   I. Lepot, "Complements on Surrogate Based Optimization for Engineering Design," vol. 41, no. 1, pp. 1–12, 2009.

[7]   A. TOURLIDAKIS, "Numerical Modelling Of Viscous Turbomachinery Flows With a Pressure Correction Method," Cranfield Institute of Technology, 1992.

[8]   D. Eckardt, "Instantaneous Measurements in the Jet-Wake Discharge Flow of a Centrifugal Compressor Impeller," *J. Eng. Power*, vol. 97, no. 3, pp. 337–345, 1975.

[9]   D. Eckardt, "Detailed Flow Investigations Within a High-Speed Centrifugal Compressor Impeller," *J. Fluids Eng.*, vol. 98, no. 3, pp. 390–399, 1976.

[10]   ANSYS, "Module 5: Multiple Zone Modeling." 2016.

[11]   S. Gibson, L, Galloway, L, Kim, S & Spence, "Assessment of turbulence model predictions for a centrifugal compressor simulation GPPF-2017-0138," 2017.

[12]   ANSYS, "ANSYS CFX Reference Guide," *Elements*, vol. 15317, no. November. pp. 724–746, 2011.