Journal of Solid and Fluid Mechanics (JSFM), 14(3): 1-4, 2024

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: 10.22044/jsfm.2024.14264.3844

Manufacturing Engineering of Iran

Numerical study of the performance parameters of the developed centrifugal compressor using a classical machine learning algorithm

Amir Hossein Aliyan[1], Hosein Mahdavy moghaddam[2,*]

[1] MSc student, Department of Aerospace Engineering, University K.N.Toosi University of Technology, Tehran, Iran

[2] Assoc. Prof., Department of Aerospace Engineering, University K.N.Toosi University of Technology, Tehran, Iran

*Corresponding author: mahdavy@kntu.ac.ir

Received: 03/09/2024 Revised: 09/09/2024 Accepted: 10/02/2024

Abstract

In recent years, significant progress has been made in research related to turbomachine. Nowadays, due to the importance of optimization and finding the optimal model, economic components and reducing the weight of the structure in each stage of compressors are considered. Due to the use of these devices in air engines, regular and diesel engines, and use in petro refineries, it can be counted among its importance. Due to the commercial nature of these data, research studies in this field are less clearly visible. The approach of the current research is to reduce the number of blades of a compressor and compensate for this reduction by changing blade angles with the help of a classical machine learning algorithm. In the next step, the analysis of the flow in the new geometry has been investigated with the Ansys CFX software, which is based on the finite element numerical method. In fact, with this work, a kind of verification has been done. In line with this research, the effect of changing the number of blades on functional parameters such as entropy and pressure ratio was evaluated. It was observed that it is possible to reduce the weight of the structure by reducing the number of blades, which reduces the pressure ratio by 5%. In addition, this research proved that the classic machine learning algorithms still have the ability to compete with more advanced algorithms. For turbulence modeling, $\boldsymbol{k-\varepsilon\ sst}$ model is used. This model has been used to accurately observe the reverse flow in the downstream flow of the compressor.

Keywords: Radial compressor; Diesel and air engine; Turbocharge; Optimization; Numerical analysis.

1. Introduction

The aim of this research is to investigate the parametric optimization criteria of a radial compressor. In the following, an overview of recent research related to this topic will be discussed.

First, Maroni [1] proposed an optimal compressor model for a system including a heat pump. The model was developed using an optimized algorithm, based on thermodynamic parameters and centerline layout. The model's validity was evaluated by testing five working fluids. A heat pump was then integrated into the system to assess its impact on cycle efficiency. Results showed that increased cycle efficiency, due to higher compressor capacity, led to a noticeable decrease in exergy.

Keylong [2] employed a multi-parameter genetic algorithm to optimize compressor pressure ratio and efficiency. The algorithm used a time-step numerical method for flowline analysis and was applied to two types of compressors. The method proved effective for compressors with higher pressure ratios. Performance mapping revealed a 2.27% increase in pressure ratio and a 0.5% improvement in isentropic efficiency. Acoustic analysis confirmed the compressor's stability, contributing to increased service life and reduced structural fatigue.

Geometric optimizations considered changes in effective components. Rokhshan [3] simulated an optimal surge to reduce pollutant emissions. The study employed a geometry modification approach, altering blade height, base and tip locations, and separator blade positioning. Without significant changes to the overall geometry, compressor capacity increased by 15%, while maintaining the pressure ratio. Combustion-related issues highlighted the impact of combustion product temperature on emissions, requiring further investigation.

Jianting Song [4] enhanced compressor efficiency and performance by injecting moisture into the air and validating the experimental model. Moisture injection lowered fluid temperature during compression, as droplet pressure reached vapor pressure, causing phase changes. This resulted in a decrease in average fluid temperature. Compared to no injection, moisture injection increased power and efficiency at similar compression ratios. The study analyzed the effects of injected particle diameter and moisture flow rate, finding that increasing these parameters led to a 4% power increase. An optimal injection diameter of approximately 10 micrometers was identified. Finally, a performance map was presented for the surge and stall range of the proposed model.

Abbasi et al. [5] conducted simulations on a row of CC3 compressor vanes to evaluate performance from stagnation to choking. Pressure and velocity variations along the compressor and flow rates were analyzed. The results demonstrated that efficiency decreases at stagnation flow due to recirculating vortices and flow reversal. At the operating flow, efficiency and pressure ratio reached their maximum, followed by a decrease due to reduced outlet static pressure caused by increased outlet velocity approaching sonic speed and choking. In choke flow, both efficiency and pressure ratio decreased with reduced outlet static pressure. This research provided valuable insights for numerical analysis and design optimization.

Ji Li [6] similarly implemented and numerically investigated optimization results on a surge.

Gary Skakh et al [7] conducted laboratory studies on the impact of improved stability on compressor performance. They found that adjusting diffuser placement angles within a ±8-degree range could enhance relative performance in a compressor with a 4:1 pressure ratio.

Jason Burgios et al [8] explored various turbulence models to simulate compressor flow. The k-ε sst model emerged as the most optimal for simulating compressor fluid flow, accurately predicting flow reversal at critical points. Energy cascade analysis further supported these findings.

Laboratory studies by Skakh et al.

Zheng Yuan et al. [9] investigated the optimal operating point of a compressor within a specific working regime. They found that reducing the Reynolds number by 70% led to a 1% decrease in pressure ratio and delayed boundary layer growth, which was acceptable for the desired pressure ratios.

The primary objective of this study is to employ a classical machine learning algorithm, gradient descent, to optimize the geometry of a CC3 compressor by modifying the number of blades.

A numerical analysis will then be conducted to evaluate the effectiveness of this algorithm.

The following outcomes are anticipated:

- **Algorithm Efficiency:** The classic gradient descent algorithm will be demonstrated to be effective in comparison to more complex algorithms like neural networks and genetic algorithms. Its simplicity and reduced computational requirements make it a faster and more cost-effective choice.
- **Geometric Optimization:** A modified geometric structure will be achieved, leading to a reduction in weight. If this modification is verified to be effective, it will offer economic benefits.

2. Methodology

The following steps were undertaken in this research:

1. **Optimal Geometry Generation:** The original CC3 geometry was modified to create an optimized configuration.
2. **Implementation of Gradient Descent Algorithm:** The optimized geometry was generated using the classic gradient descent machine learning algorithm.
3. **CFD Analysis:** To validate the accuracy of the machine learning results, a numerical analysis was conducted. The results were then compared and verified against both the original and improvedgeometries as documented in other sources.

Below are some relationships that have been used in this research:

First, the relations of gradient descent algorithm:

$$f_{\vec{w}_i,b}(\vec{x}) = \vec{w_i}^T . \vec{x_i} + b \qquad (1)$$

$$\vec{w} = \begin{bmatrix} w_1 \\ w_2 \\ \vdots \\ w_n \end{bmatrix} \vec{x} = \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{bmatrix}$$
$$J(w,b) = \frac{1}{2m}\sum_{i=1}^{m}(f_{\vec{w}_i,b}(\vec{x}) - y)^2 \qquad (2)$$

$$w_{new} = w_{old} - \alpha \frac{\partial}{\partial w} J(w,b)$$
$$b_{new} = b_{old} - \alpha \frac{\partial}{\partial b} J(w,b) \qquad (3)$$

In this research, blade angles, blade root position, and blade number were considered as three variables (x_1, x_2, and x_3) [10, 11].

The points where the surge and stall phenomena occur in a specific cycle or efficiency were also included as condition (b) (Relation 1-3).

The super-output gradient reduction code generated a new geometry through vane reduction. Numerical analysis revealed that optimal design can be achieved by reducing the number of blades and adjusting the design angles of blades in a single-stage compressor.

Furthermore, the numerical analysis validated the following propositions:

1. **Algorithm Accuracy:** The classical gradient descent algorithm demonstrated comparable performance to advanced algorithms like neural networks and genetics for this compressor model.
2. **Verification Sufficiency:** Numerical analysis alone is a suitable verification method for machine outputs, as confirmed by similar studies. However, further research will involve validating the numerical results against reliable sources.

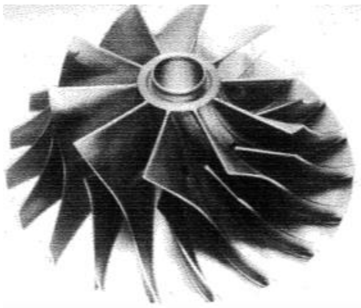

Figure 1 - CC3 original sample

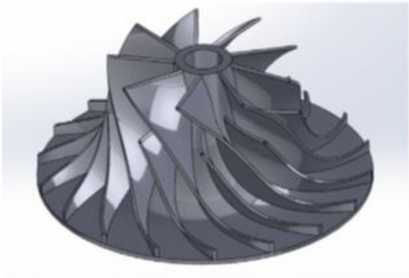

Figure 2- Geometry designed for study

Figure 1 shows the original CC3 geometry and Figure 2 shows the optimized geometry under study.
In the next step, the relations of the governing physics will be discussed:
As noted, continuity, momentum and energy equations (4-6) have been used in this research. Also, for a compressible fluid, relation (7 and 8) was used. The prerequisite of these relations is the elementary relations of the speed triangle. It should be noted that these equations have been used in the settings of the solver with the finite element numerical method (4-dimensional mesh) [12 and 13].
For the equation of continuity and momentum of any working fluid, the following relations are established (reservation form):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \quad (4)$$

$$\frac{\partial \rho u_j}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial}{\partial x_j}(2S_{ij} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}) \quad (5)$$

The energy equation used is as follows:

$$dH = dE + d(P\forall) = C_P dT \quad (6)$$

For an ideal compressible fluid that works in the regime 0.3 <M and for each spatial step, the following relations are established:

$$\frac{P_t}{P} = (1 - \eta_s \frac{\gamma - 1}{2} M^2)^{\frac{\gamma}{\gamma - 1}} \quad (7)$$

$$\frac{T_t}{T} = (1 - \eta_s \frac{\gamma - 1}{2} M^2) \quad (8)$$

Also, to calculate the velocity and Mach number of the fluid, to calculate the entropy, and for the mass flow rate passing through the surface A for the flow line at each location, the following relationships have been evaluated:

$$M = \frac{v}{a} \quad (9)$$

$$a = \sqrt{\gamma R T} \quad (10)$$

$$Tds = du + Pdv \quad (11)$$

$$du = c_v dT \quad \therefore ds = \frac{1}{T}(c_v dT + Pdv) \quad (12)$$

$$\dot{m} = \rho_0 A a_{01} (\frac{T}{T_{01}})^{\frac{\gamma+1}{2(\gamma-1)}} \quad (13)$$

3. Mesh generation

At this step, it is necessary to perform meshing with the TURBO GRID tool. In the new attack edge geometry, ELIPS is used instead of CUT OFF mode. In the same way, the reverse flow is prevented by reducing the number of vanes.

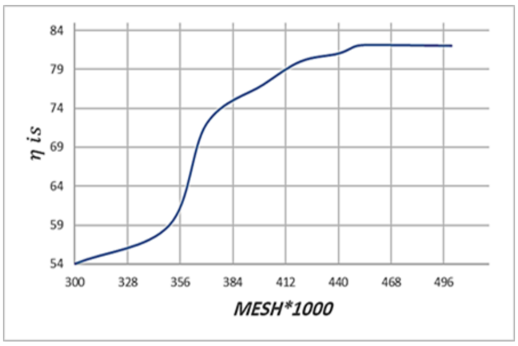

Figure 3- Mesh generated in grid-independent areas

Fig. 3 showes that the results become slightly different after 450000 grid size (grid-independent). Finally, the new geometry was studied with the number of 450234 meshes.

4. Results and Discussion

In this research, it was found that by reducing the number of vanes according to the machine estimation, there is no problem with the flow and the results have an acceptable measurement accuracy, although it faces

a drop in the pressure ratio of about 5% (compared to the original sample).
Next, in Figure 4, the speed contour, which is the final result of this research, is given.

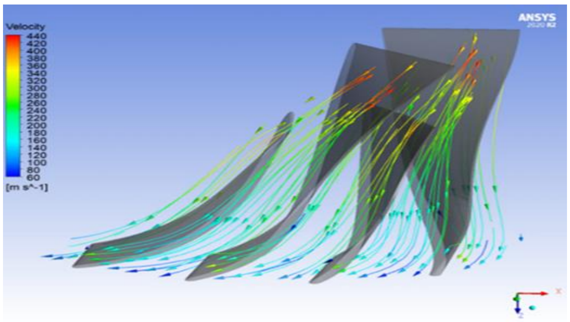

Figure 4-Velosity Contour[16]

5. Conclusions

In this research, it was observed that with a 4-digit decrease in the number of vanes, as well as a change in the return angle of the vane, there was a pressure ratio drop of about 5% (unfavorable effect). But this geometrical change had a favorable effect on the performance parameters.

As a result, it made this drop seem insignificant. The flow rate of passing fluid (increase in mass per time unit) increased by 21.2% compared to the original sample. This caused an increase in the range of suffocation in the design round. Another point is that, if the number of blades decreased more than this, there would be no way to compensate for the surge and stall phenomena. On the other hand, the increase in the number of blades also caused premature stalling in the design cycles, which requires more and more detailed investigation in this area [14]. Since it followed the weight reduction of the structure (desired effect), each vane constitutes something between 2.5% and 6.4% by weight of the compressor [14,12, 15]. As a result, it reduces the weight of the compressor by 10%. On the other hand, because the fluid has less contact with the structure, according to observations, the temperature of the fluid at the outlet decreased. This has two results: one is to increase the life of the thermal load and the other is to increase the efficiency (by changing the enthalpy ratio).

Compared to the similar sample, this sample has a significant decrease in entropy (about 7%), which is also important in terms of increasing efficiency and reducing exergy (in order to increase productivity and energy consumption). Regarding the Mach number, it is necessary to mention that the Mach of the tip of the blade is increasing up to 1.2 at the exit. But the Mach of the fluid itself decreased as expected.

6. References

[1] AndreaMeroni, Benjamin Zühls dorf, Brian Elmegaard, Fredrik Haglind (2018), Design of centrifugal compressors for heat pump systems, *Applied Energy*,Volume 232.

[2] Tang,Q.,Wu,H.Li,J.etal.(2022),Performanc Optimization of Centrifugal Compressors Based on Throughflow Model. *Arab J Sci Eng* 47,16439–16450.

[۳] رخشان، رضوان ومددی، علی وبرومند، مسعود (۱۴۰۱)، شبیه سازی ومقیاس کردن کمپرسورگریز از مرکز توربوشارژر یک موتوراحتراق داخلی رقیق سوز با روش بازطراحی یک بعدی و سه بعدی و ایجاد اصلاحات هندسی درپروانه. *مجله تحقیقات موتور*، تهران.

[4] Jianting Sun, Zhitao Zuo,Qi Liang,Xuehui Zhang, Huan Guo, Haisheng Chen. (2022), Theoretical and experimental study on effects of wet compression on centrifugal compressor performance,*Applied Thermal Engineering*,Volume 212.

[۵] عباسی، ثارالله و بغدادی محمد (۱۴۰۰)، بررسی عملکرد کمپرسور گریز از مرکز در نقاط واماندگی، *کارکرد و خفگی،بیست و نهمین همایش سالانه بین المللی انجمن مهندسان مکانیک ایران و هشتمین همایش صنعت نیروگاه‌های حرارتی*، تهران.

[6] Li,J. ,Yin,Y, Li, S. etal. (2013), Numerical simulation investigation on centrifugal compressor performance of turbocharger. *J Mech Sci Technol* 27, 1597–1601.

[7] Skoch, G. J. (2003). Experimental investigation of centrifugal compressor stabilization techniques. J., 125(9), 704-715.

[8] Assessment of turbulence model predictions for an aero-engine centrifugal compressor. *Journal of Turbomachinery*, 133(1).

[9] Lin, Yun & Gan, BinLin & Zhuge, Weilin & Zhang, Yangjun. (2013). Effects of Reynolds number on the performance of a high pressure-ratio turbocharger compressor. Science China *Technological Sciences*. 56. 10.1007/s11431-013-5213-6.

[10] Di Song, Junxian Shen, Tianchi Ma, Feiyun Xu. (2023), Multi-objective acoustic sensor placement optimization for crack detection of compressor blade based on reinforcement learning, *Mechanical Systems and Signal Processing*,Volume 197, 110350,ISSN 0888-3270.

[11] Python Scikit learn lib.,https://scikit-learn.org/stable

[12] Principles of Turbomachinery; Seppo A.Korpela, 2nd Edition.

[13] Yunus A Cengel, Michael A Boles, *Thermodynamics an Engineering Approach edition* 8.

[14] Skoch, G. J. (2003). Experimental investigation of centrifugal compressor stabilization techniques. *J. Turbomach*., 125(4), 704-713.

[15] Skoch, G. J., Prahst, P. S., Wernet, M. P., Wood, J. R., Strazisar, A.J.(1997,March).Laser Anemometer measurements of the flow field in a 4: 1pressure ratio centrifugal impeller. In ASME 1997 *International Gas Turbine and Aeroengine Congress and Exhibition*. American Society of Mechanical Engineers Digital Collection.

[16] Ansys (2020) version R2, *Turbomachinery module*.