ISSN: 1231-4005
e-ISSN: 23540133
DOI: 10.5604/01.3001.0010.3067

INFLUENCE OF SELECTED PARAMETERS ON MICRO GAS TURBINE COMPRESSOR DESIGN

Michal Czarnecki

Rzeszow University of Technology
Department of Aircraft and Aircraft Engines
Powstancow Warszawy Av. 8, 35-959 Rzeszow, Poland
tel.:+48 17 8651609, fax: +48 17 8543116
e-mail: czarn@prz.edu.pl

John Olsen

School of Mechanical and Manufacturing Engineering
The University of New South Wales
Sydney, Ainsworth Building, N.S.W., 2052, Sydney, Australia
tel.: +61 2 9385 5217, fax: +61 2 9663 1222
e-mail: j.olsen@unsw.edu.au

Abstract

The design a micro gas turbine engine is a process that requires analysis of a number of parameters. The initial stage requires consideration of more than 40 parameters [3]. The whole analysis can be made with analytical tools. However, these kinds of tools are limited to preliminary designs. After 1D-calculations and the establishment of the first CAD model, it is recommended to identify the sensitivity of the design. With a modern numerical environment such as ANSYS CFX, it is possible to predict a trend that gives the designer a 3D feedback about the initial design behaviour. For presented centrifugal compressor case, the selected parameters are vaneless diffuser space, design angle and number of stator blades. For qualitative evaluation – important results that influence design are mass flow rate, total pressure and isentropic efficiency. These results are important to turbojet engine performance and efficiency. All chosen parameters respond to given criteria. Validation and verification is still required due numerical errors that are included in CFD modelling. The advantage of 3D prediction is the possibility to eliminate gross errors before parts are sent into production.

Keywords: *jet engine, gas turbine, design, trend prediction, centrifugal compressor*

1. Introduction

The design of the compressor stage for micro jet engine starts from selecting compressor wheel from automotive parts. Most of modern designs derive from two main designs. The first is a MW54 turbojet engine developed by Wren [6]. The Wren design is characterized by compressor's external diameter of 54 mm – Garret T25 compressor wheel. The second design is related to the KJ-66 engine of German engineer Kurt Schreckling [7]. The KJ series engine is based on a 66 mm KKK K26 compressor wheel. There are modifications of this design using a TB04 compressor – 70 mm design [3]. In this article, the compressor from the GT60 turbojet is analysed in terms of influence for certain modification. The GT60 turbojet is equipped with KKK OEM 5324-123-2017 compressor wheel [4, 9].

For numerical investigation selected parameters are:
- x_3 – vaneless diffuser height coefficient,
- α_2 – diffuser blade design angle,
- n_{stat} – number of stator blades.

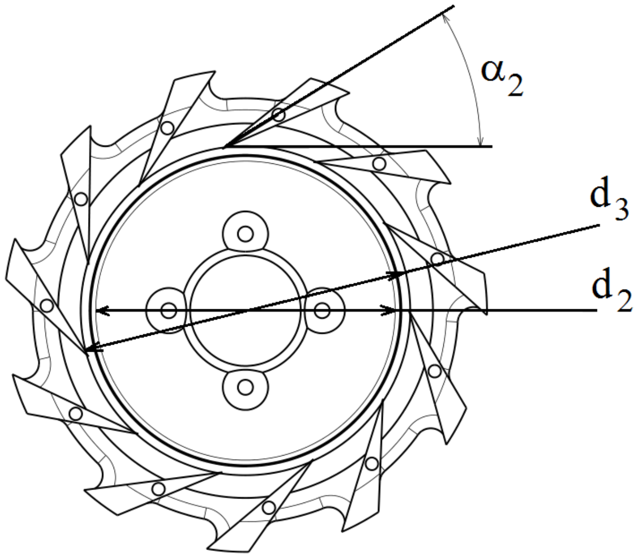

Fig. 1. Micro turbo jet engine diffuser characteristic dimensions

The vaneless diffuser height x_3 is a parameter that determines vaneless space between compressor impeller and vane diffuser. This parameter is related to external compressor dimensions by equation:

$$d_3 = x_3 \cdot d_2, \tag{1}$$

where:
d_2 – centrifugal compressor impeller external diameter.

This is a critical area in terms of diffuser operation due a range of operation from subsonic to transonic speeds. The diffuser blade of the α_{2d} design angle is an additional angle that in analytical methods is added to α_2 for compressor operation and stability improvement. Usually diffuser design of angle range is up to ± 1 deg.

The number of stator blades is a parameter that influences the diffuser in terms of available flow area and compressor efficiency. This parameter should be considered in terms of the general assembly requirements due the technology over constraint.

The selected environment for calculation was ANSYS CFX. The numerical model setup is typical for centrifugal compressor simulation. It consists of intake, rotor and stator subdomains. There are a total number of 438 knodes in the fluid subdomain: 80 knodes – intake, 165 knodes – rotor, 193 knodes – stator [4]. The selected turbulence model is an industry standard k-ε model with wall function [1, 4, 8]. This is a consequence of license limitation that available number of nodes is 512 knodes. The simulation is steady state with "stage" coupling between rotational and non-rotational part.

2. Vaneless diffuser height

The diffuser height coefficient for a micro gas turbine design is limited from a value of 1.03 up to a value of 1.09 [3, 6, 7]. For further investigations, two cases are selected. The first is 1.0745, which corresponds to initial design conditions, while the second is 1.0475, which is a reduced value. The benefit of the reduced value is to lower the vaneless diffuser and reduce the α_d blade design angle.

Comparing the velocity profile, in reference to Mach number, reducing the x_3 – parameter requires attention to the velocity profile (see Fig. 2a and 2b). The low value of a parameter results in a reduction in the maximum velocity at the exit from the diffuser. The disadvantage is a higher velocity at the entrance to the diffusion channel (Fig. 2a).

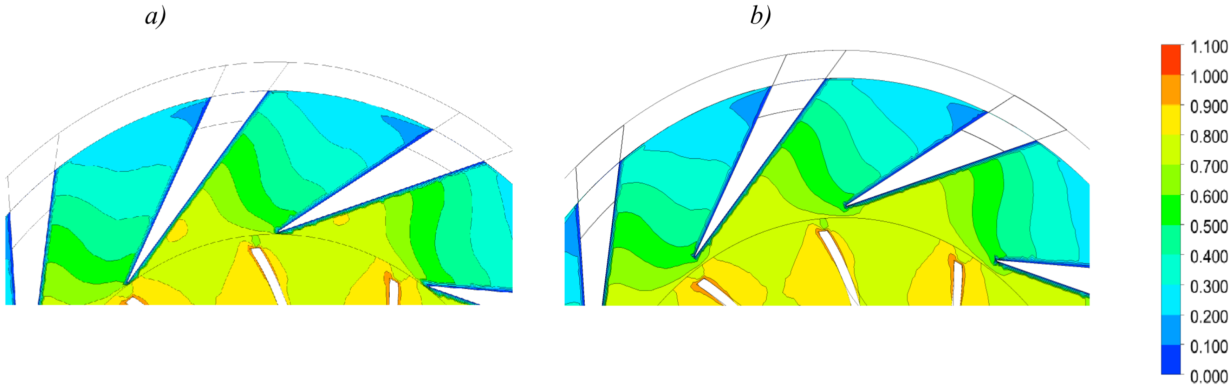

Fig. 2. Vanless diffuser space effect on a Mach number. a) Value of x_3 coefficient 1.0475, b) value of x_3 coefficient 1.0745

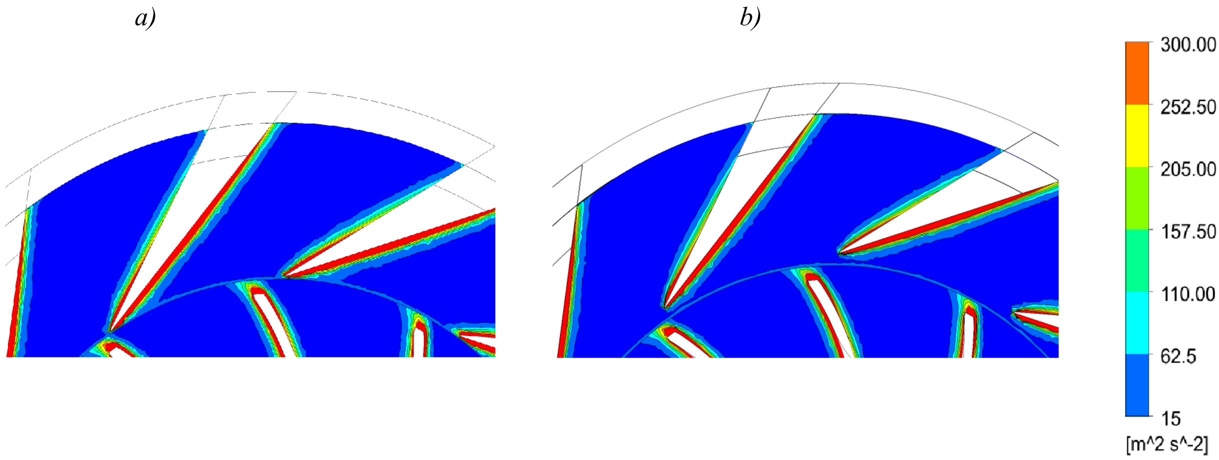

Fig. 3. Vanless diffuser space effect on turbulence intensity, a) value of x_3 coefficient 1.0475, b) value of x_3 coefficient 1.0745

In terms of turbulence coefficient, the value of the turbulence intensity is reasonable (Fig. 3a and 3b). The quality of boundary layer is correct. To obtain proper results in terms of quantity it is required to compare numerical data with the experiment.

3. Design blade angle

At the design of centrifugal diffuser is to add to blade design angle α_2 additional ±0.5°, which refers to analytical design point angle (ADP). In terms of numerical simulation design conditions was have been preserved. The main aspect was to identify influence modification for design performance. The presentation of the results in graphical form was omitted due to the comparative scale, which was used for considerations. In general, comparison to ADP simulation, adding design angle increases for mass flow rate by 0.01%. Main benefit is increasing of isentropic efficiency by 0.6%.

4. Number of blades

Another important parameter is a number of blades in bladed diffuser. In a large-scale commercially operated centrifugal compressor, the number of blades is recommended to be from 16 up to 36 [2, 5]. In a microscale design, the designer is limited by the available space due to reduced frontal diameter and the required assembly conditions etc. [3]. For the numerical simulations, three cases are selected, they being: 11, 13, 15 blades (Fig. 4).

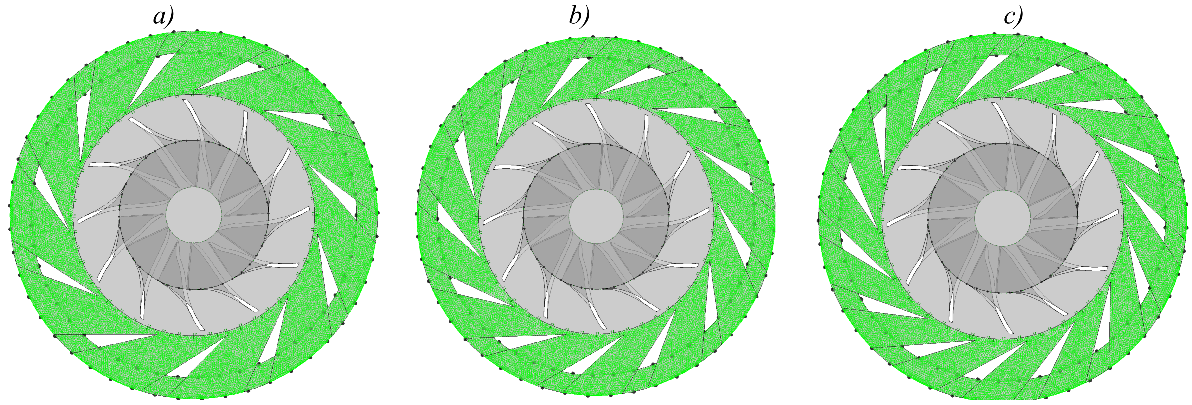

Fig. 4. Competitive design a) 11 blades, b) 13 blades, c) 15 blades

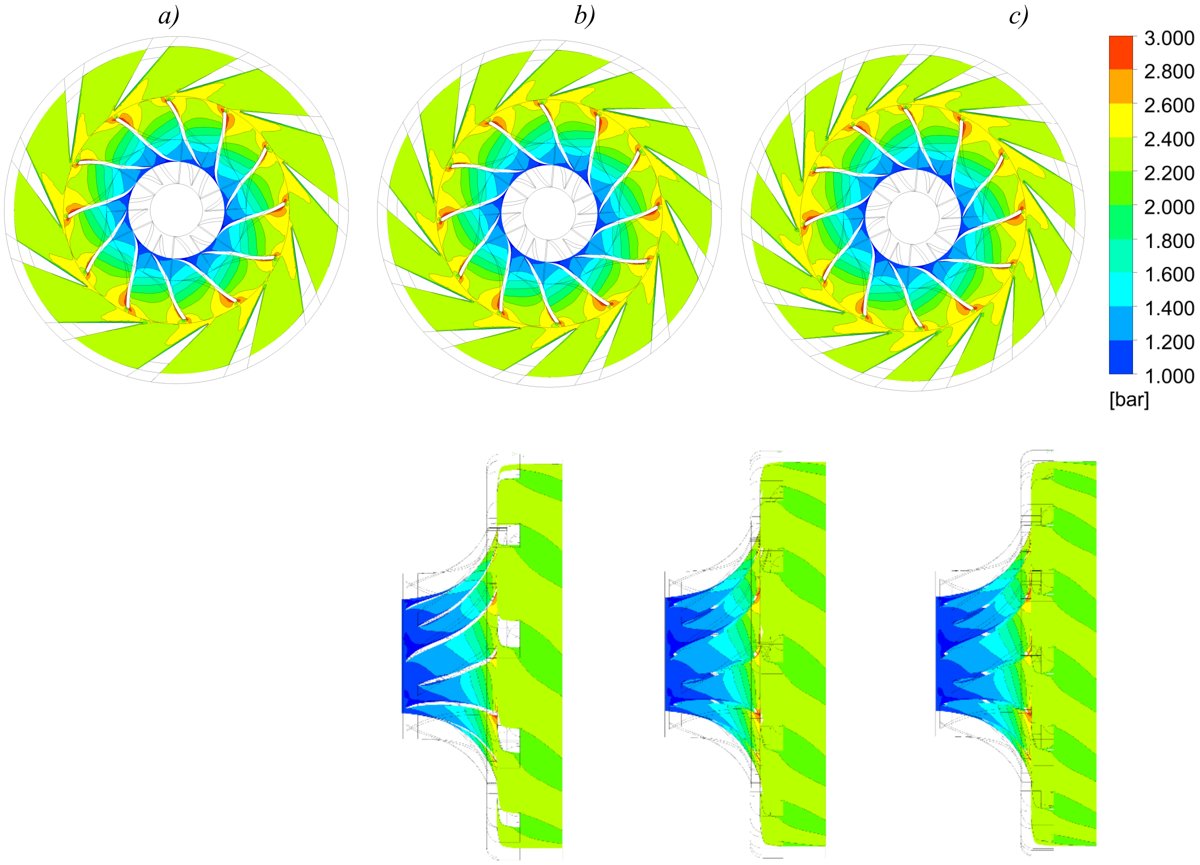

Fig. 5. Total pressure distribution a) 11 blades, b) 13 blades, c) 15 blades

An unpaired number of blades are chosen to avoid resonance between the impeller and the diffuser. Note that the number of rotor blades is 12. The designed diffuser blades are staged at α_2 = 67.5 deg, while the coefficient for vaneless diffuser is x_3 = 1.0745. Some selected design cases are presented below. In terms of quality of total pressure distribution, for the selected cases – Fig. 5, the pressure distribution in the rotor – stator conditions are even and stable. Due to relatively low increment in radial direction for a vaneless diffuser, it is possible to estimate the number of diffuser blades that do not require numerical simulation to estimate flow conditions in vaneless and vane diffuser. Only the Mach number should be considered to avoid a high supersonic value.

$$n_{dif} \leq \left(\frac{d_3 \cdot h_s}{d_2 \cdot h_r}\right) \cdot n_{imp}, \tag{2}$$

where:
d_3 – compressor diffuser external dimension,
h_s – compressor stator flow channel height,
h_r – compressor rotor flow channel height,
n_{imp} – number of impeller blades.

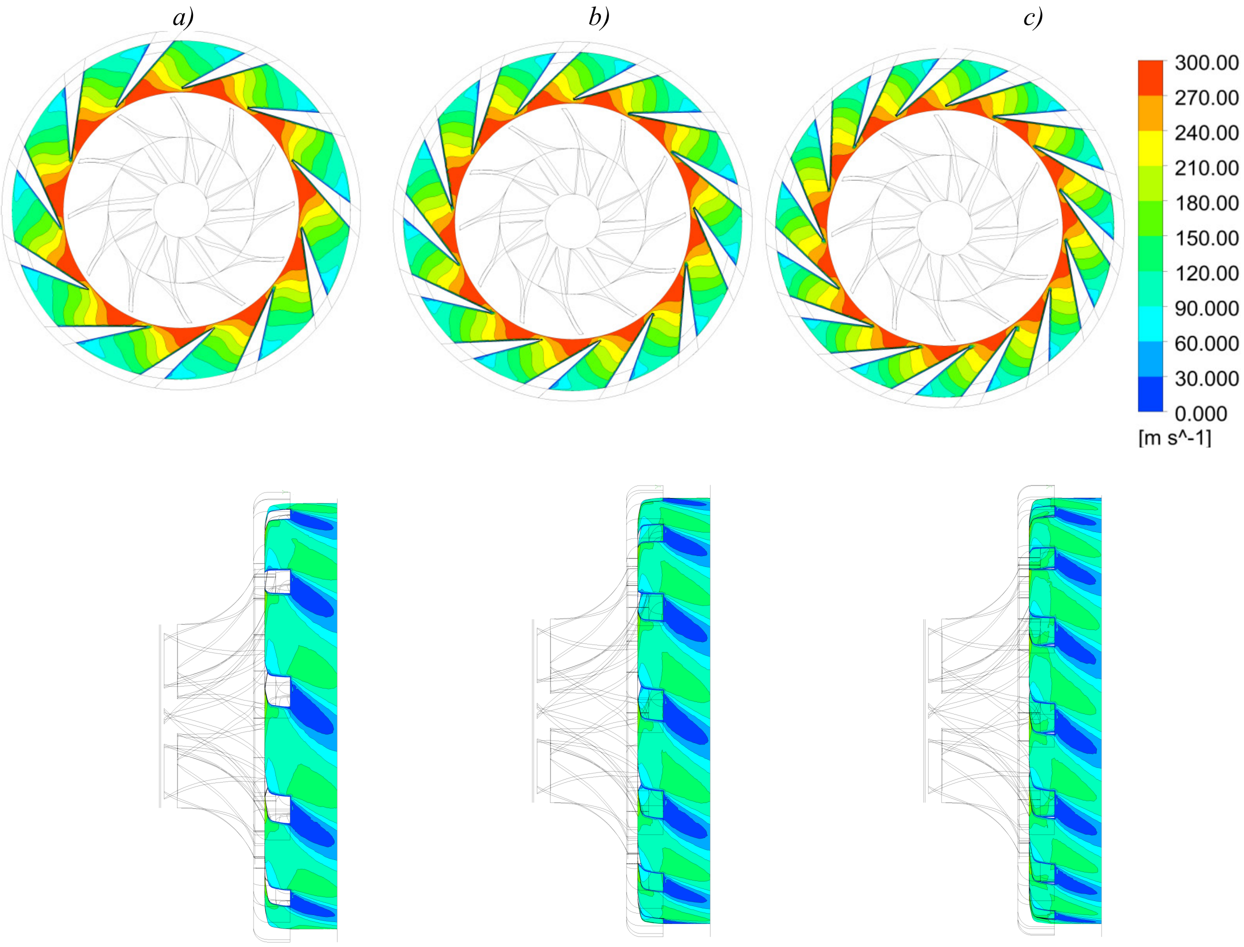

Fig. 6. Velocity distribution in diffuser area a) 11 blades, b) 13 blades, c) 15 blades

The isentropic efficiency of a compressor stage was also reduced due to increasing of the number of vanes (Fig. 6). The results are for variant a – 0.7076, variant b – 0.7065, variant c – 0.7042. In percentage efficiency, the penalty dropped in comparison to reference design was 0.115% – 13-blade case, 0.482% – 15-blade case. Efficiency penalty is a result of increased friction between fluid and stator surface. An additional disadvantage is a highly turbulent flow after the diffuser in all cases vanes geometry modification is required to reduce turbulence and increase overall efficiency (Fig. 6, 7).

5. Design modification

In reference to the design cases, a relatively simple modification was proposed to improve the design. The main result was to reduce turbulence at the exit of a diffuser (Fig. 7). In comparison to design case presented at Fig. 5, the improved design has a slightly reduced aerodynamic trail after the stator vane (Fig. 8). An additional effect is an improvement in the pressure distribution on the high-pressure side of stator vane. The rotor/stator interaction in terms of quality of operation is similar to previous design.

Comparing the velocity distribution to the design case (Fig. 6), the distribution can be seen, there is a reduced speed at the exit from the diffuser that is beneficial in terms of stage pressure recovery (Fig. 9). A major advantage of the proposed modification is the reduction in turbulence at the exit from the stator. The reduced turbulence intensity provides information about friction lose in the channel (Fig. 10). The modified version has a beneficial reduction of turbulence intensity of approximately 30%.

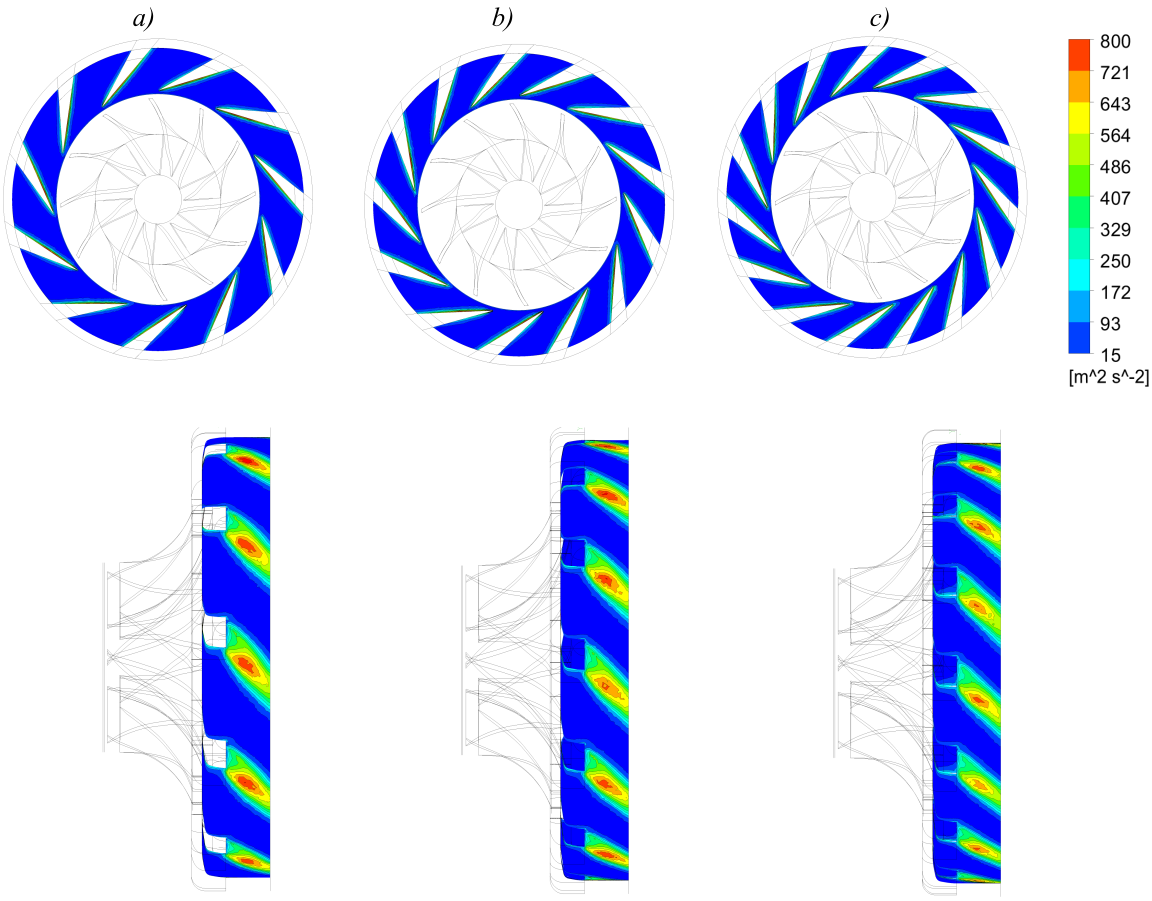

Fig. 7. Turbulence distribution in diffuser area a) 11 blades, b) 13 blades, c) 15 blades

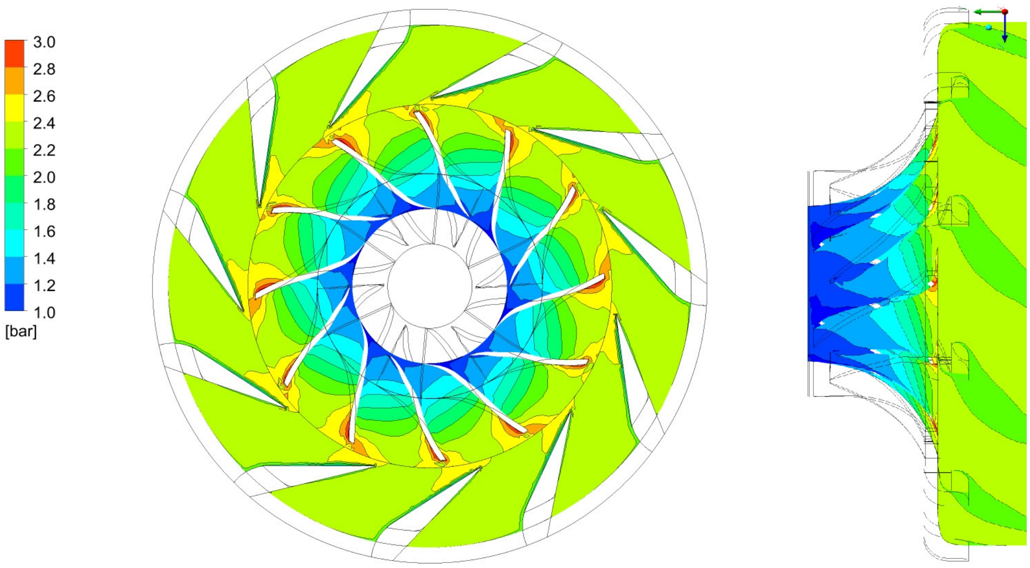

Fig. 8. Total pressure distribution in modified design

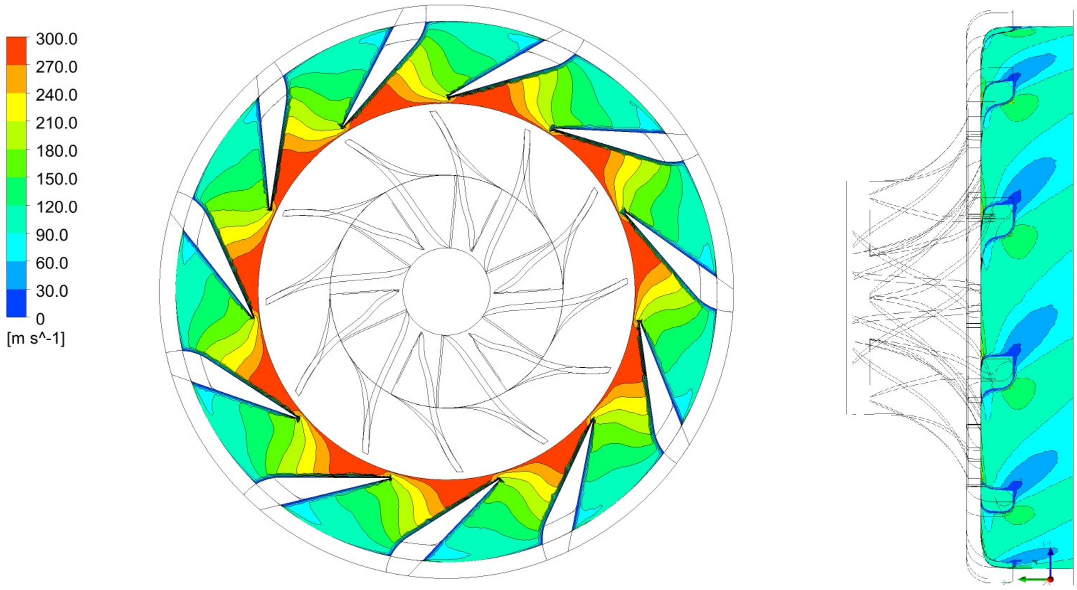

Fig. 9. Velocity profile distribution in modified design

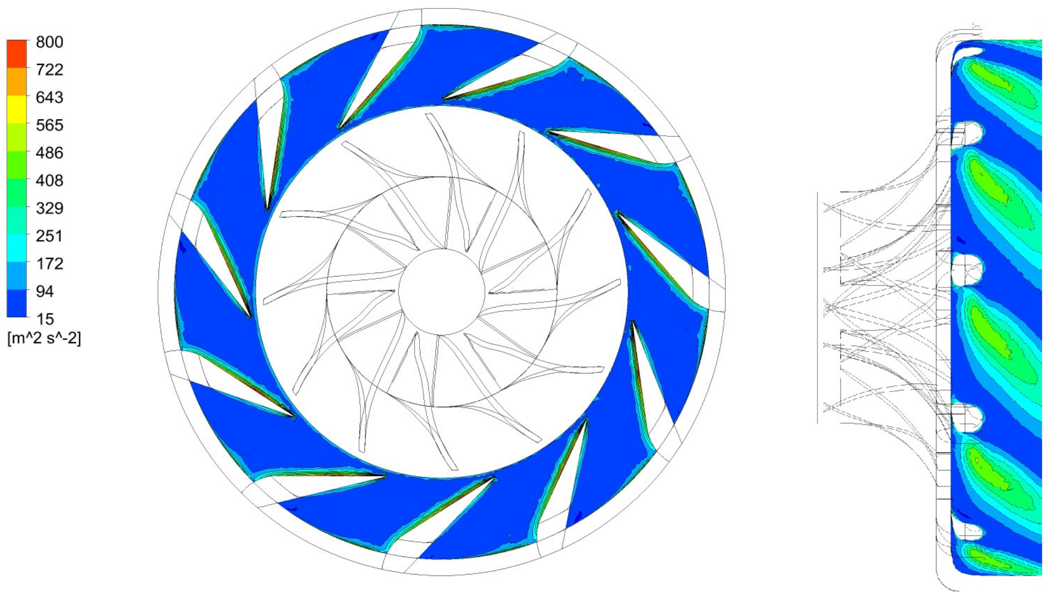

Fig. 10. Modified design – turbulence intensity.

6. Summary

The presented numerical simulations were the results of a design study. At this level of design, the industry standard *k-ε* model for turbulence was applied. This model long with the obtained results will be compared with a real engine that will be built and tested. The advantage of analytical methods over computational fluid dynamics is simplicity and low cost in terms of resources. The advantage of using a computational fluid dynamics over an analytical method is that the three dimensional environment allows the designer to rate a design from its internal behaviour. At this conceptual stage, some numerical inaccuracy of the computational model is accepted [8]. The use of computational fluid mechanics is considered as a tool to assist and predict the product behaviour only.

Final valuation for design cases was performed by indicator system. Selected indicators are:

M – mass flow rate (engine performance),
P – total pressure at the exit (engine performance, efficiency of the design),
E – isentropic efficiency (efficiency of the design, fuel consumption).

A positive mark was considered an improvement over base design case (11BL_10745). Markings and a result was presented at Tab. 1.

Tab. 1. Centrifugal compressor diffusers indicators

Test case	Design aim	Airflow	Total pressure	Isentropic efficiency	Trend		
		m_{air} [kg/s]	p_{2tt} [bar]	η_{ctt}	M	P	E
11BL_10745		0.16641	2.189678	0.70760	base case		
11BL_1045	Diffuser height effect	0.166572	2.185798	0.7073	+	-	-
11BL_67	Design angle consideration	0.166797	2.188417	0.7082	+	-	+
11BL_68		0.166779	2.1883588	0.7082	+	-	+
13BL_10745	Number of blades effect	0.165654	2.187059	0.7065	-	-	-
15BL_10745		0.164601	2.184149	0.7042	-	-	-
MOD	Stator modification	0.16577	2.226	0.70598	-	+	-

For selected case, it was found:

Problem A – Diffuser height effect. Test case: 11BL_10745, 11BL_104,

– reducing diffuser height has positive impact on mass flow rate of the engine,
– has positive change in terms of technology of a gas turbine (blade shape);

Problem B – Design angle consideration – Test case: 11BL-10745, 11BL_67, 11BL_68:

– adding additional 0.5 deg to α_2 change benefits in mass flow and stage efficiency,
– obtained result need to be verified in real design conditions;

Problem C – Number of blades effect – Test case: 11BL-10745, 13BL_10745, 15BL_10745:

– due a technological over constrain of the design is not recommended to design with more than 13 blades,
– increase number of stator blades decrease all indicators;

Problem D – Stator modification:

– modification improves performance of the design.

Acknowledgement

Presented work was an effect of a PANTHER research project, which was awarded under Erasmus Mundus Action II by EU.

References

[1] Avaramenko, A., Frolov, A., Hamalainen, *Simulation and modelling of flow in gas compressor*, Journal of Applied Mathematics, Vol. 2015.
[2] Cohen, H., Rodgers, G. F. C., Saravanamutto, H. I. H., *Gas turbine theory,* Longman Group limited, Essex, 1996.
[3] Czarnecki, M., *Design and experimental research of microgastrubine turbojet engine*, Rzeszow 2013
[4] Czarnecki, M, *Micro scale gas turbine diffuser design – a practical approach,* Journal of Applied Fluid Mechanics.
[5] Dixon, S. L., Hall, C. A., *Fluid mechanics and thermodynamics of turbomachinery*, Elsevier, Tokyo 2010.
[6] Parish, R., Wright, A., Murphy, M., *Plans for the MW54 gas turbine second edition*, Wren turbines LTD., August 2000.
[7] Schreckling, K., *Gas Turbine Engines for Model Aircraft*, Traplet Publications Limited, Worcestershire 2003
[8] Tu, J. Y., Yeoh, G. H., Liu, C., *Computational Fluid Dynamics: A practical approach*, Elsevier, 2012.
[9] BW*, Performance turbocharger catalog 2012 Edition* (BorgWarner Turbo Systems).