IOP Conf. Series: Materials Science and Engineering **90** (2015) 012046 doi:10.1088/1757-899X/90/1/012046

Multi-objective optimization for impeller shroud contour, the width of vane diffuser and the number of blades of the centrifugal compressor stage based on the CFD calculation

A M Danilishin, Y V Kozhukhov, V K Yun

Department of Compressor, Vacuum and Refrigeration Engineering, Peter the Great St. Petersburg Polytechnic University, 195251 St. Petersburg, Russia, Politechnicheskaya St., 29

E-mail: lelliaa@mail.ru

Abstract. The study presents the results of multi-objective optimization for a high flow centrifugal compressor stage with impeller pressure ratio Π=1.3 and conditional flow rate coefficient Φ=0.108. The compressor stage includes an impeller with 3D backswept blades, a vane diffuser and an axial inlet. The goal of optimization is to increase polytropic efficiency and polytropic head ratio of the basic design. The CFD method is used to estimate compressor efficiency at rated duty. CAE optimization is applied based on the parametric optimization of impeller shroud contour and number of blades of the impeller and diffuser versus the original design. Parameterization of geometrics is also used to the width-ratio of vane diffuser in the range of b_3/D_2= 0.08-0.097. The study considers 52 cases of the optimization of impeller shroud contour and the number of blades in the search for the improved design. The optimization procedure uses the automatic generated computation grid and supplementary activation of solution to each design case. The numerical calculation for each case has been performed automatically by ANSYS CFX 14.5 soft application. The optimization allowed to obtain the improved design with total polytropic efficiency increase by 1.58% for the impeller and polytropic head coefficient increase by 0.58%. The polytropic efficiency and the polytropic head coefficient are calculated on the total parameters. The performance of the basic impeller has been exhaustibly validated by test records provided JSC "REPH ZAO". The resulted error range does not exceed 5% over the performance map, except near surge point.

Legend

b [-] dimensionless width
D [-] dimensionless diameter
h [$J \cdot kg^{-1}$] head
$\dot{m}$ [$kg \cdot s^{-1}$] mass flow
n [-] mode number
P [Pa] pressure
T [K] temperature
U [$m \cdot s^{-1}$] peripheral velocity
X [mm] driving dimension in axial direction
y^+ [-] wall distance in normal direction
Y [mm] driving dimension in radial direction
Z [-] number of blades
Π [-] total to total pressure ratio
β [-] loss coefficient
γ [-] performance parameter
ρ [$kg \cdot m^{-3}$] density
δ [-] relative error
η [-] efficiency coefficient
ψ[-] head coefficient
Φ[-] conditional flow rate coefficient

Subscripts

0, 1, 2,... - specified section
2,3 -blade element (impeller, diffuser)
p, d, i - polytropic, dynamic, internal
lk. , d.fr. - leak, disk friction
inl., out - inlet, outlet

Superscripts

*- total parameters

1. Introduction

Centrifugal compressors are widespread in all main industries, and they are also used for production, processing, transportation and storage of natural gas. It is obvious that regardless of the service of the centrifugal compressor, the power consumption is estimated in thousands of kilowatts, which increases the requirements for the unit energy efficiency and encourages consumers and producers to find the ways to increase it.

Nowadays, the most commonly used methods for centrifugal compressor flow parts design are fluid dynamics methods (CFD). It is a standard way using for optimization of the flow part to increase its efficiency.

The scientific problem lies in the area of timesaving as the developing of simulation models for the CFD optimization is time-consuming due to the fact that the designer is required to calculate a few dozens or even hundreds variants of flow parts of centrifugal compressor stages to achieve the specified requirements of energy efficiency. This problem can be solved by the application of multi-objective optimization methods that automatically carry out a series of calculations with the use of supercomputing technology.

Multi-objective optimization allows to simultaneously search for cases of flow part by such important parameters (or criteria) as efficiency, pressure ratio, head, design constraints, etc ..

The above information confirms the need to develop the optimization procedure and to set forth the purposes of study:

- To validate CFD calculations of the basic design with experimental data.
- To define geometric parameters and energy efficiency criteria of the centrifugal compressor flow part for optimization.
- To develop the procedures for automatic optimization of the shroud contour, the width of vane diffuser and the number of blades for the centrifugal compressor flow part based on CFD.

2. Subject of research

The subject of research is a high flow centrifugal compressor stage model (Fig. 1), consisting of axial inlet nozzle, impeller with 3D backswept blades and vane diffuser designed by Nevsky Zavod (JSC "REPH ZAO") against advanced parameter of total to total pressure ratio Π= 1.3 in impeller for pipeline compressors of natural gas. The main dimensionless geometrical parameters of the flow part are provided in Table. 1.

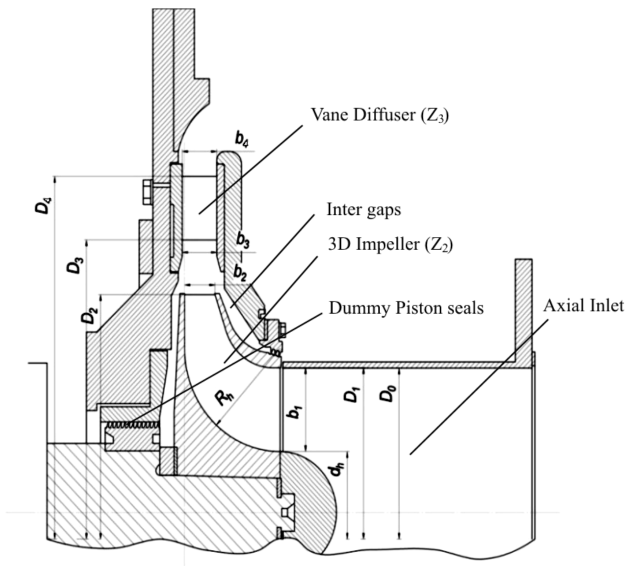

Fig. 1 Scheme of the object of study

Table 1. The basic dimensionless parameters of stage.

Parameter	Value
D_0	0.663
D_1	0.664
D_h	0.28
D_3	1.25
D_4	1.54
b_1	0.192
b_2	0.076
b_3	0.086
b_4	0.086
R_h	0.237
Z_2, pcs.	24
Z_3, pcs.	11

3. CFD model

CFD model has one vaned item with periodic boundary conditions. The collector chamber, inter gaps and dummy piston seals have not been simulated. The calculation is based on High-Re turbulence models considering the wall functions relevant to the flow near the wall at a relatively coarse computational grid, which allows to reduce the time period needed for simulation. Therefore, the 1st grid node should be in the logarithmic boundary layer ($y^+ = 30 ... 300$). A CFD Solver of Ansys CFX 14.5 was applied. The calculation was performed by averaging the Navier-Stokes equations RANS (Reynolds-averaged Navier-Stokes) for models of turbulence: Spalart-Allmaras (SA), RNG k-ε (RNG), Shear Stress Transport (SST). In this case SST turbulence model involves the use of a standard k-ε turbulence model, this is effected by the blending function depending on the wall distance. The service fluid is air (ideal gas).

The simulation of a large number of cases takes a lot of time hence, to shorten time necessary for calculations is one of the priorities. The main factor influencing the duration of each operation of calculation is the size of the computational grid, therefore, the first task was to find a computational grid with the minimum possible number of cells which adequately describes the physics of the flow. The outcome of the mesh independence studies provides the grid with 259275 cells.

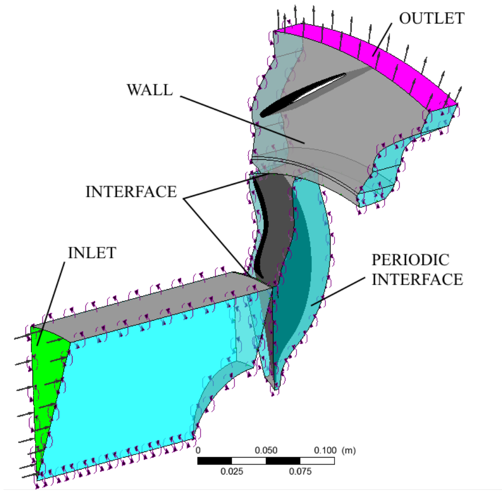

Fig. 2. CFD model and BL conditions of centrifugal compressor stage

4. Battery limits conditions

The inflow direction is assumed as normal to the axial inlet battery limits (BL) within grid computational domain. The simulations were run with P^*_{inl} and T^*_{inl} in inlet boundary and mass flow $\dot{m}_{out}$ at outlet boundary. The walls were assumed as adiabatic. The inlet and outlet BL conditions are listed in Table. 2. The Table also provides the conditional coefficient of flow rate (Φ), inlet total density (ρ^*_{inl}) and stage mass flow ($\dot{m}$). The BL conditions were determined by the outcomes of the full-scale experiment. An upwind discretization scheme of 2nd order was included into solver settings. The grid interface ('stage') is used for rotor-stator grid connection, which averaged flow parameters in the circumference direction.

Table 2. The parameters set on the inlet and outlet BL of the computational domain

Mode n	Φ	T^*_{inl}, K	P^*_{inl}, Pa	ρ^*_{inl}, $kg{\cdot}m^{-3}$	$\dot{m}$, $kg{\cdot}s^{-1}$	$\dot{m}_{out}$, $kg{\cdot}s^{-1}$
1	0.1491	291.53	90800	1.080	2.880	0.26181
2	0.1437	291.63	91500	1.088	2.796	0.25418
3	0.1369	291.73	92300	1.097	2.686	0.24418
4	0.1281	291.73	93200	1.108	2.539	0.23078
5	0.1195	291.87	94200	1.119	2.393	0.21754
6	0.1084	291.82	95300	1.132	2.197	0.19971
7	0.0946	292.06	96600	1.147	1.942	0.17655

5. Validation results

All modes developed the converging solutions (example of convergence history for mode 6 shown in Fig. 3), defined by the following parameters within the computational domain BL:

- the fall of the level of root-mean-square (RMS) residuals below 10^{-5},
- the relative error in mass imbalance is $(1\text{-}3) \cdot 10^{-3}$ %,
- the relative error in the energy imbalance is $(1\text{-}5) \cdot 10^{-3}$ %

The controlled values of integral parameters in the control sections do not change from iteration to iteration. The control of calculation accuracy is provided by means of validation with experimental data provided by JSC "REPH ZAO". Validation results are shown in Fig. 4. Hereinafter point №1 refers to the maximum flow mode, point №7 refers to the near surge point.

According to the experimental data, mode 7 has the maximum value of the sum of the coefficients of friction disk and leaks $\beta_{lk.} + \beta_{d.fr.} = 0.0165$; while mode №1 provides minimum sum $\beta_{lk.} + \beta_{d.fr.} = 0.0113$. These values are typical for high flow centrifugal compressors, which is described in detail in the literature [4]. The earlier studies [6] for middle flow 2D impeller with Φ range 0.06 - 0.09 and $\beta_{lk.} + \beta_{d.fr.}$ (0.0208-1.0146) showed that exclusion of inter gaps and seal from the computational domain increased the efficiency by 2.2 % average. Therefore, taking into account that for high flow centrifugal compressors stage the influence of such items is even less, we can assume that the efficiency increase in subject will not more than 2.2%.

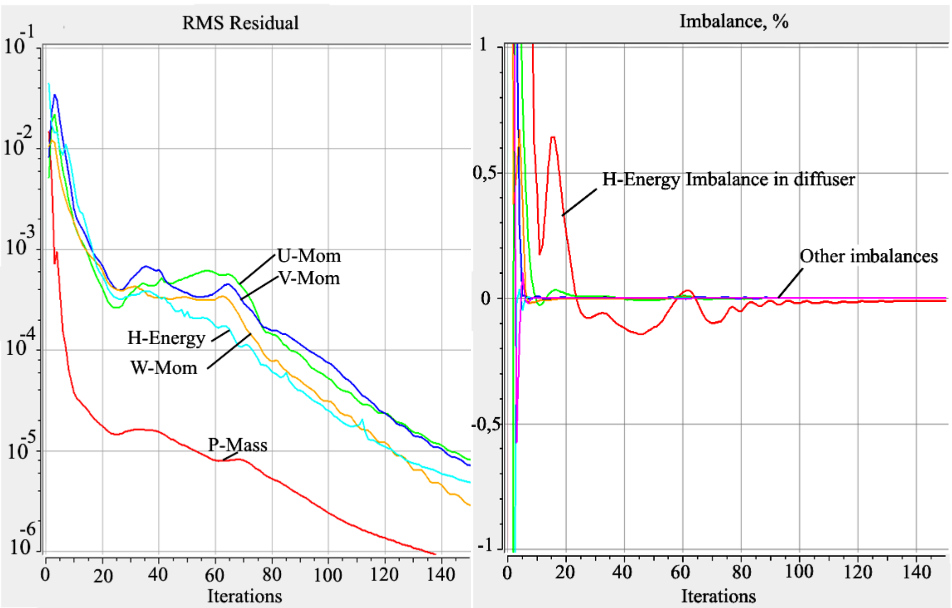

Fig. 3. Convergence history for RMS residuals mass and momentum, energy and imbalances within computations domains

Conditional flow rate coefficient is

$$\Phi = \frac{4 \cdot \dot{m}}{\rho_{inl}^{*} \pi D_2^2 U_2} \tag{1}$$

Polytropic efficiency at full range of parameters is

$$\eta_p^* = {h_p^*}/{h_i}, \quad h_p^* = h_p + h_d. \tag{2}$$

Polytropic head coefficient at full range of parameters is

$$\psi_p^* = h_p^* / u_2^2 \tag{3}$$

Total to total pressure ratio coefficient

$$\Pi = P_{out}^* \Big/ P_{inl}^* \tag{4}$$

Mean relative error for n-modes throughout the performance

$$\delta_{mean} = \frac{1}{n}\sum_{i=1}^{n}\left|\frac{\gamma_i - \gamma_{\text{iexperiment}}}{\gamma_{\text{iexperiment}}}\right| \tag{5}$$

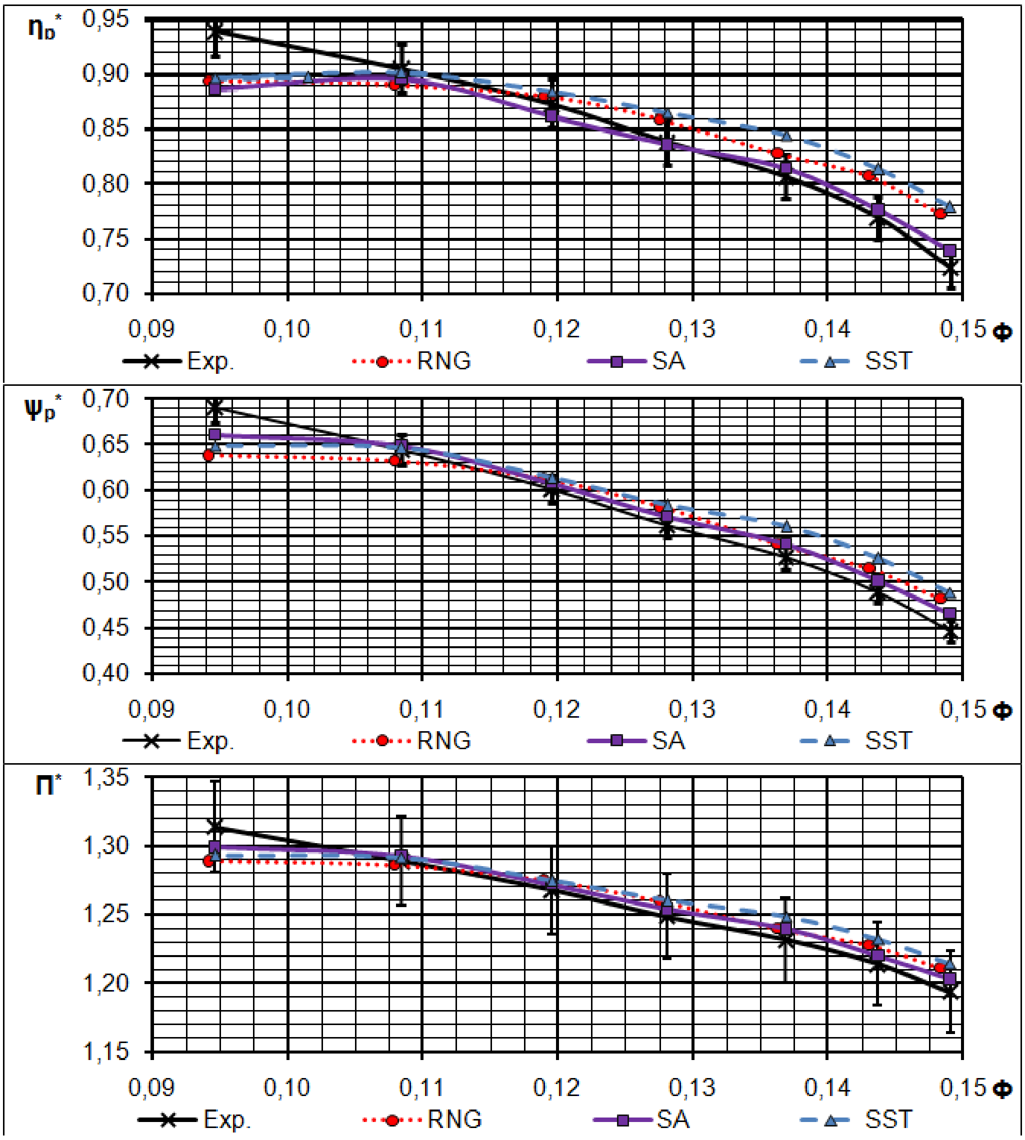

Fig. 4. Validation results for three turbulence models at the outlet of the impeller

Spalart-Allmaras turbulence model showed good accuracy with experiment and the received error does not exceed the level of engineering errors 5%, except near surge point that illustrated on fig. 4. Also it showed the best accuracy throughout the all performance parameters η_p^*, ψ_p^* and Π as shown in table 3. Using the formula (5), Table 3 shows the mean relative error δ_{mean} of 7- modes throughout the all performance graphs for each solution turbulence model. In the formula (5) is determined by the relative error performance parameters with respect to the experimental data, and then averaged by the number of the modes. Considering the said above, we selected the turbulence model Spalart-Allmaras (SA) for the further optimization.

Table 3. Mean relative error for 7- modes throughout the all performance graphs for each model turbulence solution.

	δ_{mean}		
Model turbulence:	η_p^*	ψ_p^*	Π
SST	3,9%	5,1%	1,1%
RNG	3,5%	4,4%	1,0%
SA	1,8%	2,4%	0,6%

6. The optimization procedure

The optimization procedure was developed by a soft tool Ansys Workbench 14.5 with the help of software Ansys Design Exploration v 14.5. the optimization was performed for the design mode №6. Figure 5 shows the optimization procedure carried out by Ansys Workbench 14.5 (left) and parameterization diagram (right) of the shroud contour centreline of the CC impeller. The same kind of parameterization was carried out for vane diffuser and inlet nozzle. All soft tools, including the grid generator and the solver were interfaced, which allows to adjust the computational domain automatically and to calculate the cases based on the input data.

Considering the design criteria one should remember that is better to minimize their number as it will lead to the increased amount of new cases and, consequently, solver actuations. It is also necessary to set optimization parameters adequately, that is why their values were discretely entered by hand. This approach allows to avoid a number of uncontrolled cases.

The optimization procedure divided into two steps. The first task is to optimize the shroud contour and width of vane diffuser. The shape is controlled by regulated dimensions in axial and radial direction, called X and Y (see fig. 5, right pic.).The hub contour is kept constant. The impeller shroud contour is adjusted so that it is remained lower than basic case. The dimensionless width of vane diffuser (b_3) is changed four times from $8 \cdot 10^{-2}$ to $9.7 \cdot 10^{-2}$. Regulated dimensions are set in a discrete mode for X and Y: 0, 4, 8. Eventually we come to three optimization parameters. The total number of cases is estimated as $(1+2^3) \cdot 4=36$. The pattern of variations for the impeller shroud contour is shown in fig. 6.

The second task is to optimize the number of blades. The number of blades is assumed as follows: Z_2=26, 24, 22, 20 for impeller blades and Z_3 = 11,13,15,17 for vane diffuser. Eventually we receive 2^4=16 cases in total.

The purpose of multi-objective optimization is to achieve the best efficiency for centrifugal compressor stage. The selected objective function of optimization for all tasks was the polytropic efficiency η_p^* and total to total pressure ratio Π on the impeller and the diffuser outlet sections. As soon as the optimization is initiated, the outputs are provided automatically (cases and grid generation).

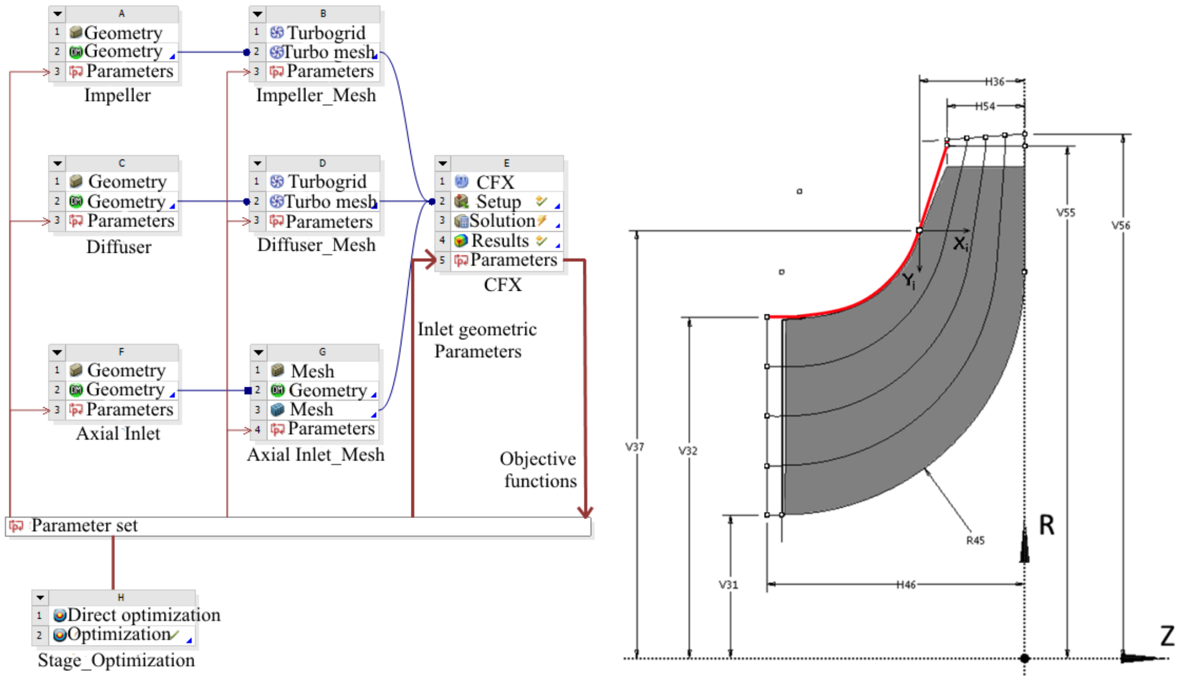

Fig. 5. The interface diagram between soft tools and the impeller shroud contour parameterization

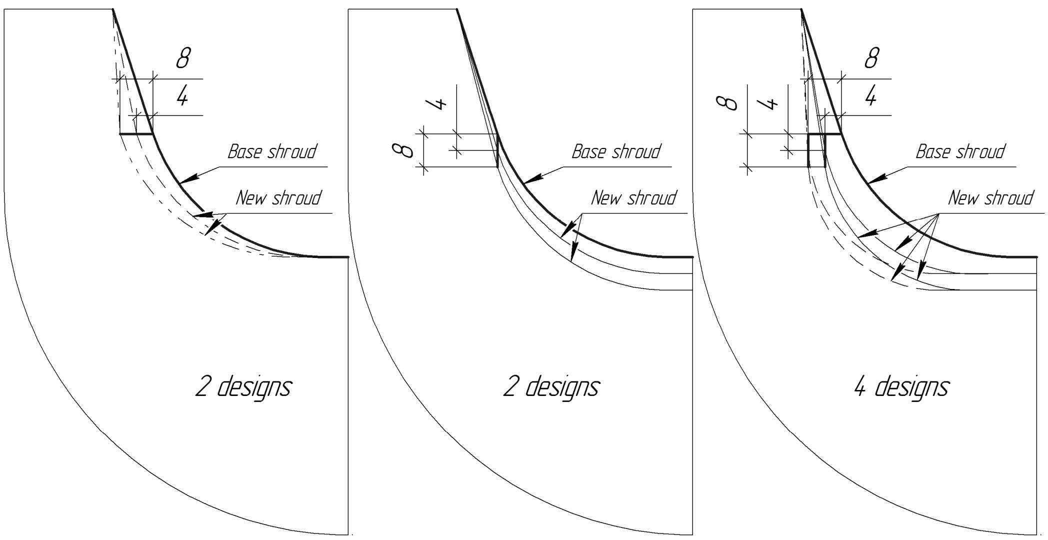

Fig. 6. The optimization cases for shroud contour of the impeller

7. Optimization results

Graphics (Fig 7 and 8) provide the results of the optimization. X-axis shows the cases for shroud contour of the impeller and number of vane diffusers. Y-axis shows η_p^* (left) and ψ_p^* (right). The review of these graphs (Fig 7,8) gives an idea of the efficiency increase of the stage versus adjustment of design parameters.

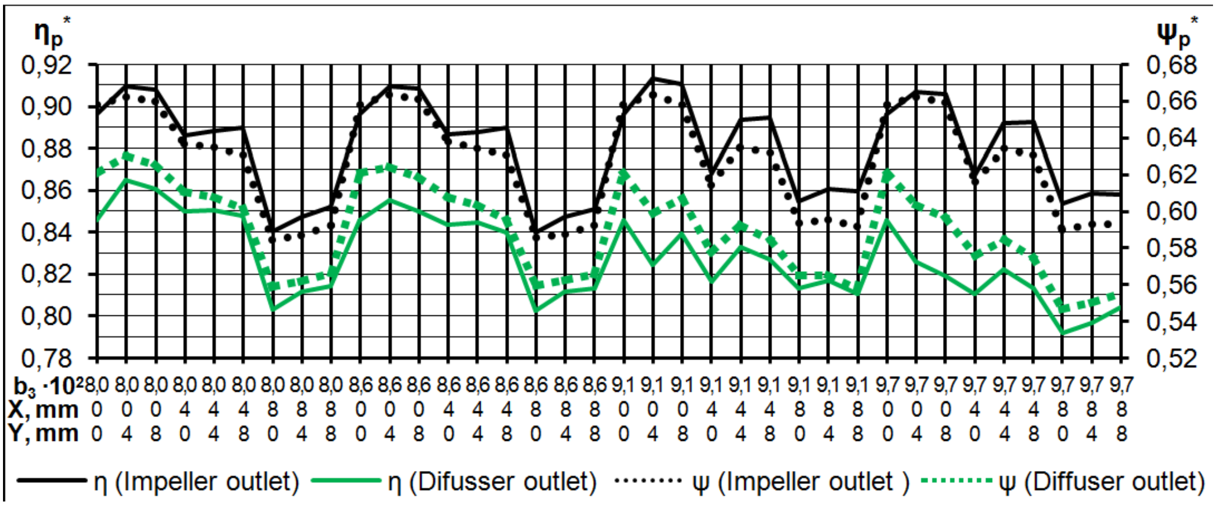

Fig. 7. The results of the optimization of the impeller shroud contour and the width of vane diffuser (b3) with controlled values X and Y.

The Optimization of impeller shroud counter showed a marked worsening of the efficiency and the head of the stage if controlled value of X is changed. The change of the width of the diffuser has little effect on the efficiency of the stage. The best result is achieved when controlled value of X = 4 or 8 mm and Y = 0, specifically η_p^* is increased by 1.33% and ψ_p by 0.57% at impeller outlet, and η_p^* by 0.96% and ψ_p^* by 0.32% at diffuser outlet, accordingly.

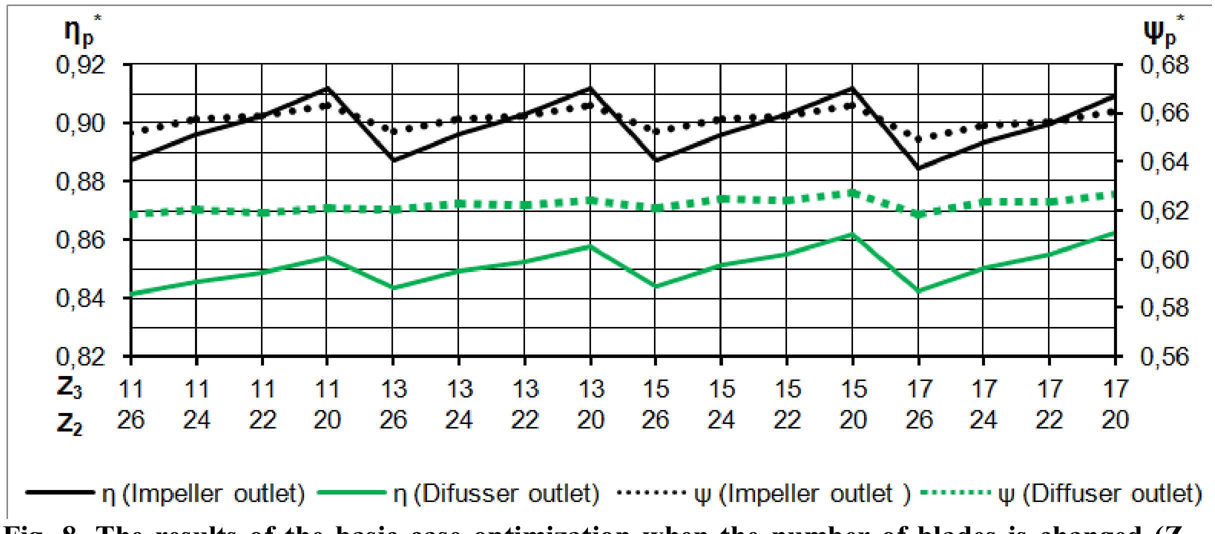

Fig. 8. The results of the basic case optimization when the number of blades is changed (Z_2 - impeller, Z_3 – diffuser)

Graph in fig. 8 shows that the reduction of the number of impeller blades from 24 to 20 increases η_p^* by 1.58% and ψ_p by 0.58%. It is justified by the less friction due to smaller work surface area. The simulation also shows that at Z_3 = 13 (number of vane diffuser blades) we have the increase of η_p^* by 1.66%.

Table 4 summarizes the best results for each of the two-step optimization (the best shroud counter optimization and the best number of blades optimization) versus the basic design. Here it is also presented the outcomes for the case where we try to combine the two best results (design parameters

X=4, Y=0, Z_2 = 20 and Z_3 = 13) of each step in attempt to obtain the best optimization. However, this idea failed to prove.

Table 4. The best optimization results v basic design.

Design	η_p^* Imp.	Change	η_p^* Dif.	Change	ψ_p^* Imp.	Change	ψ_p^* Dif.	Change
Base	0.896		0.846		0.658		0.621	
Shroud best	0.909	+1.33%	0.855	+0.96%	0.663	+0.57%	0.624	+0.32%
Z_2, Z_3 best	0.912	+1.58%	0.862	+1.66%	0.663	+0.58%	0.627	+0.65%
Both	0.911	+1.5	0.825	-2.1%	0.671	+1.3%	0.608	-1.3%

8. Conclusions

The soft tool Ansys 14.5 was used for the development of the multi-objected optimization. The subject of study for optimization is centrifugal compressor stage with impeller pressure ratio Π=1,3 and conditional flow rate coefficient Φ=0.108. The turbulence model Spalart-Allmares was selected based on the preliminary validation with experimental data. % The deviations of the design parameters from the experimental data are within 5% over the performance map.

The optimization procedure was divided into two steps:

- to optimize the shroud contour based on two controlled values and four different values of the width of vane diffuser;
- to optimize the number of blades of impeller.

There were 52 design cases received. The optimization was carried out for the design mode №6.

The best result of Step 1 is the stage with X=0, Y=4 and increase of η_p^* by 1.33%, ψ_p^* by 0.57% at the impeller outlet and increase of η_p^* by 0.96%, ψ_p^* by 0.32% at the vane diffuser outlet. The best result of Step 2 is increase of η_p^* by 1.58%, ψ_p^* by 0.58% at the impeller outlet and increase of η_p^* by 1.66%, ψ_p^* by 0.65% at the vane diffuser outlet at Z_2 = 20 and Z_3 = 13. Then we developed the case which combined the best results of two steps, but the outcome was negative, in particular the decrease η_p* by 2.1%, ψ_p* by 1.3% at the vane diffuser outlet. This shows that for the best optimization it is necessary to consider the interaction of the stage units.

The developed optimization procedure gives broad opportunities for the fast adjustment of centrifugal compressor stages to the predetermined parameters of the power efficiency.

If such optimization tasks are paralleled using supercomputers, it is possible to achieve the wide range of cases by design parameters applying finer calculation grid. In addition the outcomes can be used to develop response surface. The mathematical methods of optimization can be applied to define local and global optimum, as well as Pareto set.

It is expected to further improve the possibilities of optimization by applying mathematical methods for the design parameters (criteria) analyses.

References

[1] Y. B. Galerkin, B. I. Zykov, K. P. Seleznev. Study of the shape of the meridional section of the impeller intermediate stage centrifugal compressor. Power engineering, 1965, No. 10, p. 33-39.

[2] H. P. Dickmann. Shroud contour optimization for a turbocharger centrifugal compressor trim family/ 10th European Turbomachinery Conference, Lappeenranta, Finland; 04/2013

[3] Florian Frese, Johannes Einzinger, Johannes Will. Design optimisation of an impeller with CFD and Meta-Model of optimal Prognosis (MoP) / 10th International conference on turbochargers and turbocharging, London, England; 2012

[4] K.P. Seleznev, Y. S. Podobuev YS, S.A. Anisimov. Theory and Design of turbocompressors:

Proc. manual for schools /; Ed. K.P. Seleznev - Leningrad: Mechanical Engineering, 1968. - 406p.

[5] ANSYS Inc. ANSYS CFX 14.5: Users Manual, 2012

[6] A.M. Danilishin, Y.V. Kozhukhov Analysis of the influence on the performance of centrifugal compressor stage of the model parameters in ANSYS CFX. Week of Science SPbSTU: Proceedings of the conference with international participation. Institute of Energy and Transport Systems SPbSTU. Part 2 - St. Petersburg Polytechnic University, 2014. - p.186-189