How to cite / Как сослаться на статью:
Danilishin, A.M., Kozhukhov, Y.V., Simonov, A.M. Gas-dynamic designing and profiling complex for the two-element centrifugal compressor stage with 3D impeller (2019) AIP Conference Proceedings, 2141, № 030065. DOI:10.1063/1.5122115

Gas-Dynamic Designing And Profiling Complex For The Two-Element Centrifugal Compressor Stage With 3d Impeller

A.M. Danilishin[1a)], Y.V. Kozhukhov[1], A.M. Simonov[1]

[1]Peter the Great St.Petersburg Polytechnic University
Russia, 195251, St. Petersburg, Politehnicheskaya st., 29.

[a)]Corresponding author: Danilishin_am@mail.ru

Abstract. To carry out research, a complex of gas-dynamic design and profiling of a two-stage centrifugal compressor with an axial-radial impeller has been developed. It is a continuous process of solving the inverse and direct problem of gas-dynamic design, profiling the flow section with subsequent analysis of the rational velocity distribution of an inviscid flow on the impeller blades on axisymmetric streamline surfaces.

INTRODUCTION

The design of the flow part of a centrifugal compressor usually begins with a variant gas-dynamic calculation, which consists of two components: the solution of the inverse and direct problem. In solving the inverse problem are given: inlet parameters, pressure ratio, mass flow, optimal gas-dynamic parameters and geometric relations of the flow part. As a result, the dimensions of the flow part are determined. In the direct problem, the geometric parameters of the stage and the rotor speed are set. As a result, the flow parameters in the flow part and its efficiency are determined. The main characteristics of losses in the elements of the flow part are determined by mathematical models of losses, test data of model and full-scale stages, the experience of the designer. According to the results of the variant gas-dynamic calculation, the elements of the flow part are profiled and, if necessary, return to the inverse problem for selecting and refining the parameters of the flow part. The use of mathematical models has led to the creation of various methods for calculating losses, many of which are based on the generalization of experimental data, others are based on numerical methods for calculating flows in the flow path.

By the end of 80-ies of the widely used methods for the inviscid calculation of the flow, implemented in a program for the individual elements of the turbochargers (diffusers, blade grids , individual profiles, etc.). Among the methods we can distinguish the method of singularities, finite-difference method, finite element method, the method of finite (control) volumes. The singularity method is used to calculate the plane potential flows of an ideal fluid in multi-connected regions in which the potential (vortex-free) flow on the profile contour is replaced by a system of singularities - vortices. The method allows us to calculate the gas flow in the radial plane, which is modeled flow around the grating profiles. Finite-difference methods are characterized by the fact that in equations differentials are replaced by finite differences, and the equations themselves from partial differential equations become equations in finite differences. The approach [3] is known, in which the three-dimensional inviscid flow problem is reduced to two two-dimensional. Here the flow is considered on two families of linear current surfaces S1 and S2. The meridional flow in the S1 plane is calculated and the flow around the lattice in a layer of variable thickness on axisymmetric surfaces S2 is calculated. B. N. Savin developed a refined channel method for calculating the velocity distribution on the blades. The method allows to take into account approximately the flow lag from the blades at the outlet, the change in the velocity distribution and the load on the blades at different flow regimes and flow rate. Currently, modern methods of computational fluid dynamics (CFD) are widely used to calculate the viscous three-dimensional turbulent flow in the flow part, to obtain the gas-dynamic characteristics of centrifugal compressors and to estimate losses in the elements of the flow part. Recently, there is a growing interest in multi-criteria optimization, which is devoted to research [2, 5, 6, 7, 8] for 3D impellers. Integration of multi-objective optimization methods into CFD helps to significantly increase the efficiency of the designed stage by searching for the optimal flow part shape in the meridional and radial sections.

These methods have their advantages. Mathematical models based on experimental data allow to obtain sufficiently accurate and fast results. However, they are limited by a number of experiments included in the model. In the case of the design of advanced stages, not included in the series, can give a significant error. Numerical methods for the calculation of inviscid flow it possible to analyze the flow structure. Quantitative comparison of design variants helps to evaluate the most effective option. There are restrictions on the use of models. CFD methods in some cases give reliable qualitative and quantitative representation. The complexity and time costs limit the comprehensive use of methods due to the need to use powerful computing tools – supercomputers and highly qualified personnel. It should be recognized that the experiment is currently the only way to reliably assess the characteristics of the centrifugal compressor. However, these methods can significantly reduce, if not abandon the finishing tests. Reduce the time spent on R&D for new gas-dynamic centrifugal compressor projects.

For these purposes, the authors have developed a complex gas-dynamic design and profiling for two-element centrifugal compressor stage with 3D impeller. The complex is a step-by-step process of solving the inverse and direct problem of gas-dynamic design. Profiling and analysis of the rational distribution of relative velocities of inviscid flow on impeller blades on axisymmetric streamlines. The estimated calculation of losses in the elements of the approximate techniques and CFD methods. Step-by-step specification of loss factors for gas-dynamic calculation. At the final stage, multi-objective optimization of the flow part is performed to achieve the highest efficiency of the stage.

Figure 1 shows the General design procedure. The complex is a further development of automated methods of complex computational and theoretical research [1]. Implementation of algorithms of gas-dynamic calculation, profiling and approximate inviscid calculation is made in the program code and programs written in C++.

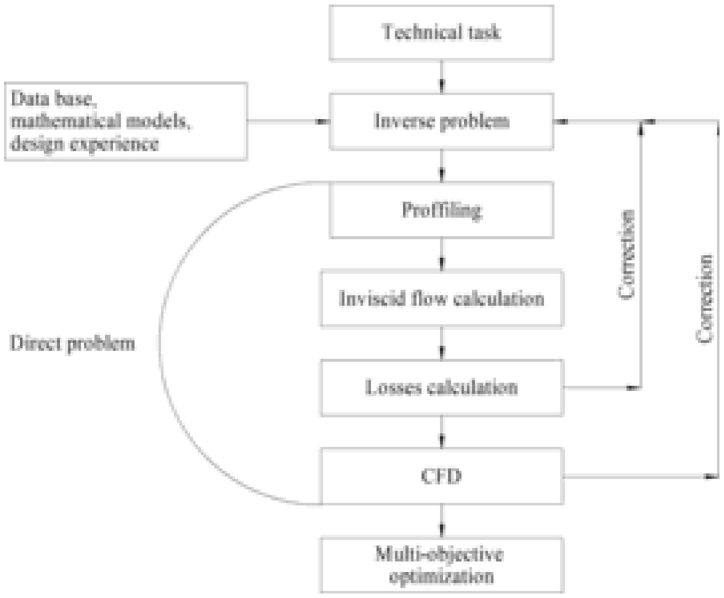

FIGURE 1. General design procedure

GAS DYNAMIC CALCULATION

Algorithms for solving the direct and inverse problem of gas dynamic calculation are based on the design techniques developed at the Department of "Compressor, vacuum and refrigeration» (CVRE) [1, 4]. At the beginning of the design, the next values are set:
conditional flow coefficient:

$$\Phi_d = \frac{4\times\overline{\mathrm{m}}}{\rho^*_{inlet}\pi \mathrm{D}_2^2 \mathrm{U}_2} = (0.06...0.1),$$

theoretical head coefficient:

$$\psi_t = h_t \;/\; u_2^2 = 0.55...0.92,$$

the hub ratio:

$$\overline{D}_{hub} = D_{hub} \,/\, D_2 = 0.22...0.35,$$

coefficient of losses in the impeller:

$$\zeta_{imp} = 0.08...0.3,$$

the diffusivity of the flow in the impeller channel

$$n_w = w_1^{'} \,/\, w_{2m}^{'} = 1.1...1.8$$

and other parameters.

As a result, the main geometrical and gas-dynamic parameters of the flow part are determined

$$U_2 = \sqrt{\frac{H_{ad}}{\psi_{ad}}},$$

where H_{ad} - adiabatic head, ψ_{ad} - the coefficient of adiabatic head.
Impeller diameter:

$$D_2 = \sqrt{\frac{4V_{inlet}}{\pi\Phi_d \varepsilon_1 U_2}},$$

the relative diameter of the shroud of the inlet:

$$\overline{D}_{1\,w_1^{'}\min} = \sqrt{\overline{D}_{hub}^2 + \sqrt[3]{\frac{2\Phi_d^2}{\varepsilon_1^2\tau_1^2}}} \quad \text{and other,}$$

The choice of the optimal combination of structural and gas-dynamic parameters of the flow part in the elements is carried out using loss models. Models of losses developed as a result of the complex theoretical and experimental work of the CVRE Department in the exploration and creation of effective high-pressure stages with 3D impellers for compressors for General industrial use [1]. Specification of losses by means of boundary layer models and CFD models allows to extend the range of application of the algorithms. This makes it possible to calculate new gas dynamic projects that are not included in the General range of recommendations of the methodology.

PROFILING

Profiling is performed with the separation of the axial and radial parts of the impeller. The algorithm of profiling of a flowing part of impellers is developed on the basis of experience of design of 3d impellers [1]. The developed profiling program provides the formation of arrays of the output data of the coordinates of the flow part on the middle, hub and shroud contours (channel width B, blades angles β_{bl}, middle line angle inclination in the meridional plane γ, radius r in the meridional plane. The data is then used to calculate the inviscid flow and construction of a 3D model of the impeller. As shown in Fig.2, hub and shroud curves are divided into two smoothly tangential arcs. The arc of the circle is divided into n equidistant points. The point coordinates are interpolated by a cubic spline. The position of the middle line of the meridional contour is determined and an orthogonal grid is constructed. The position of the middle line of the meridional contour is determined and an orthogonal grid is constructed. In the axial and radial part of the impeller, the angles of the blades on the respective contours are determined. The output data of the profiling program are input for the program of approximate calculation of the ideal gas flow in the stage of a centrifugal compressor in a quasi-three-dimensional formulation [1].

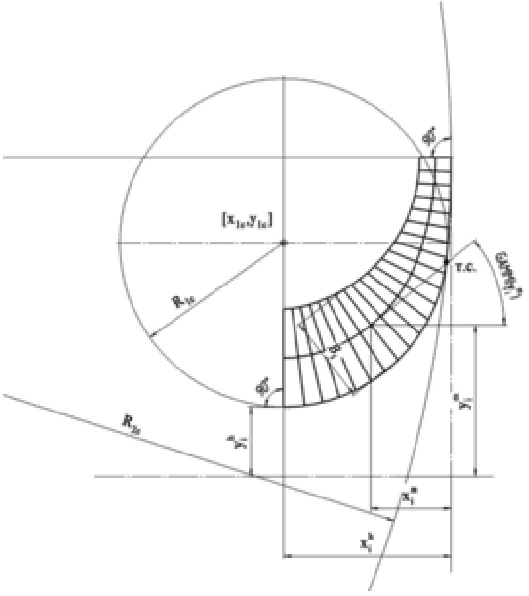

FIGURE 2. Scheme of impeller meridional profiling model

The pattern of the aerodynamic load distribution varies during the calculation process. The amount of local diffusivity on the surfaces of the blades is controlled. The velocity distributions are estimated by optimal shape patterns.

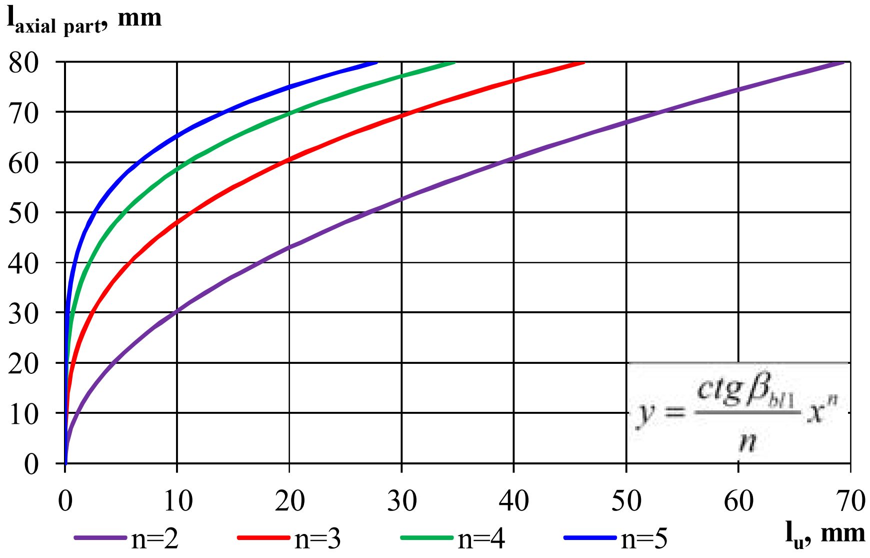

FIGURE 3. The plots of the length of the impeller axial part dependencies on the circumference direction for different shape parameters n

The profiling of the flow part model for CFD modeling is performed in the Ansys workbench software environment and integrated into the algorithm of multi-objective optimization.

INVICID AND VISCOUS CALCULATION

The main characteristic of the centrifugal impeller is the theoretical head, which determines the energy supplied to the gas by the impeller to do useful work and overcome resistance in the flow part. The existing methods for calculating the theoretical head use numerous empirical models that use the slip coefficient to account for the lag of flow from the impeller blades. [1]. In this paper, the decision was made to use the method of approximate calculation of an inviscid ideal gas flow in a centrifugal compressor stage in a quasi-three-dimensional formulation as an initial estimate of the pressure characteristic model. And to refine the results of a viscous three-dimensional calculation. The method considers the flow around a lattice lying on an axisymmetric surface of a current in a layer of variable thickness. For the impeller, relative flow is considered. The method is based on the representation of the vector of relative velocity w at an arbitrary point of the interscapular channel as a sum of two components: a vector w having the direction of the blades, and a vector Δwu, directed perpendicular to the meridional plane passing through this point: w = w + Δwu. The presence of the component Δwu is associated with the process of condensation and rarefaction of streamlines at the surfaces of the blades when they interact with the flow. At the output of the lattice, the projection vector Δwu on the axis is negative, it characterizes the lag of the flow from the blades.

The program allows you to get the optimal distribution of the velocities of an inviscid flow, which is practically ensured by repeatedly selecting the geometric parameters of the flow part: B, β_{bl}, γ. Combining the process of solving a direct gas-dynamic problem with profiling and calculating an inviscid flow allows, at the design stage, to carry out a series of variant calculations of various forms of the flow section of the impeller with determining the estimated pressure characteristics. Figure 4 shows the calculated velocity distributions of an inviscid flow for the RK-61 impeller. Viscous axisymmetric flow calculation in a vaneless diffuser based on the approximation of a narrow channel [5].

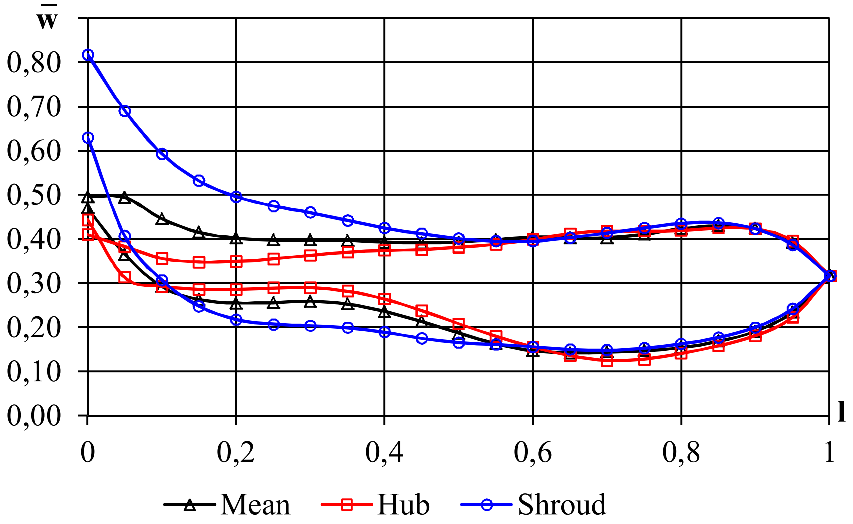

FIGURE 4. The plots of the dimensionless relative velocity distribution on the blade surface dependencies on the blade length for the model RK61 on the mean, hub and shroud streamline

Based on the velocity distribution of the invicid flow, the recommended load is chosen on the blade sections, the angle of attack is analyzed along the blade height. Then, the parameters of the boundary layer are calculated, and the values of the coefficient of loss in the impeller on the middle, hub, shroud streamlines and on the endwall surfaces are estimated. Table 1 summarizes the results of calculating the loss coefficient for the RK-61 impeller versus the experimental value.

Table 1. Profile losses coefficient distribution and comparison

Mean	**Hub**	**Shroud**	**Endwall**	**Total**	**Experiment data**
0.21	0.19	0.32	0.0138	0.25	0.20

To assess the reliability of the calculation methods used, modeling and comparison with the experiment of 12 high-pressure two-stage stages with an axial-radial impeller were performed in the Ansys CFD software package. The estimation of the relative error of modeling in the area of economical work and on the design mode was made. In the area of economical work for high-pressure two-stage stages with $\Psi t = 0.72$ in the range of flow rates $0.064 < \Phi_d < 0.1$, the maximum error for the coefficient of internal pressure (Ψi) is 3.6%, for the coefficient of theoretical pressure (Ψt) is 3.0%, for the coefficient of polytropic efficiency by full parameters (η*p) is 2.5%, for the coefficient of polytropic head in full parameters (Ψ*p) is 4.8%.Given these values, it is possible to take into account the error of CFD modeling in the design of promising stages. For variants of impellers with backswept blades, a smaller error was obtained, and for the impeller of the type “radial star” maximum. Calculations were carried out and the results of the distribution of inviscid and viscous flow in the impeller were analyzed. Satisfactory qualitative and quantitative agreement was obtained, see Fig. 5.

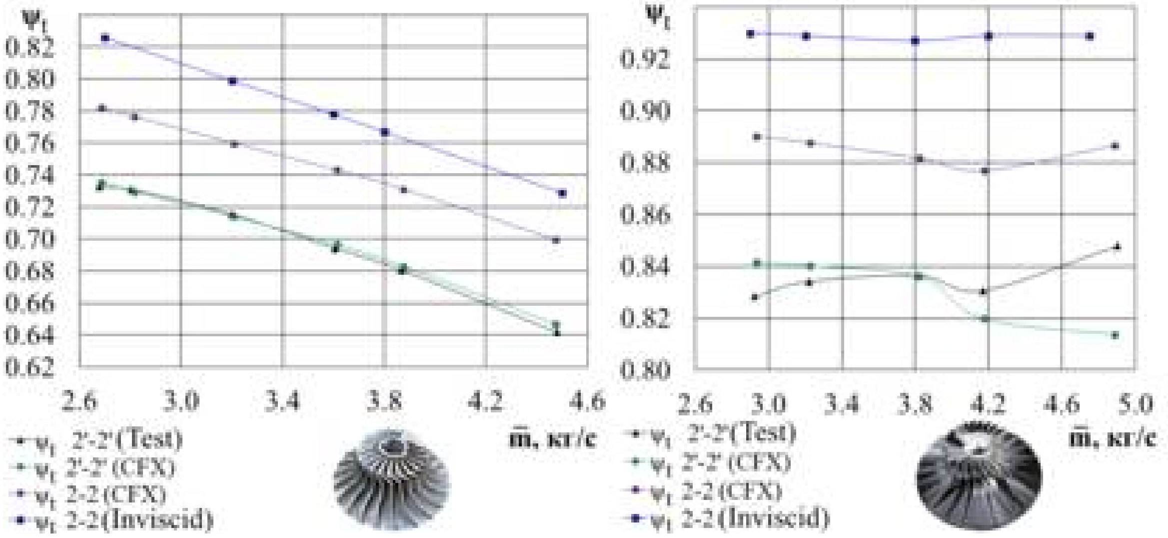

FIGURE 5. The plots of the ψ^*_t dependencies on the mass flow for the models PK61 in the section 2-2 and 2’-2’.

For RK-61 (backswept blades) in the calculation mode for t the discrepancy of the viscous calculation and the approximate non-viscous calculation program was ~ 4% in section 2-2, with a difference in viscous calculation with experiment <1% in section 2’-2 ’. For RK-51 (radial star) in the design mode for pact, the discrepancy between the viscous and non-viscous calculations was also ~ 4.0%, with the discrepancy between the viscous calculations and the experiment <1%. An inviscid calculation can be used to obtain a satisfactory characteristic of the theoretical head in an axial-radial impeller, which should reduce the total time for alternative design and development of the impeller. The final viscous calculation specifies the characteristics of the compressor, ensuring the required head values.

SAMPLE OF THE MULTI-OBJECTIVE OPTIMIZATION

For carrying out multi-criteria and multi-parametric optimization, the developed parameterized models of the flow part are used: an axial-radial impeller and a vaneless diffuser. As an example, the modernization of the impeller RC-61 is given. The following criteria were chosen as optimization criteria: maximizing the coefficient of polytropic efficiency and limiting the constant value of the coefficient of polytropic pressure Ψ*p=0.68. As parameters, the following 8 parameters were selected that ensure the change of the meridional contour: b_3/b_2=(0.8-1.6), b_4/b_2=(0.8-1.6), D_3/D_2=(1,05-1,25), z=(20-25), k_1=(0,8-1), k_2=(0,85-1,0), k_3=(1-1,3), k_z=(0.85-1).

Optimization algorithm used MOGA (Multi-Objective Genetic Algorithm). The results of the convergence of the solution of problems are presented in graph 6.

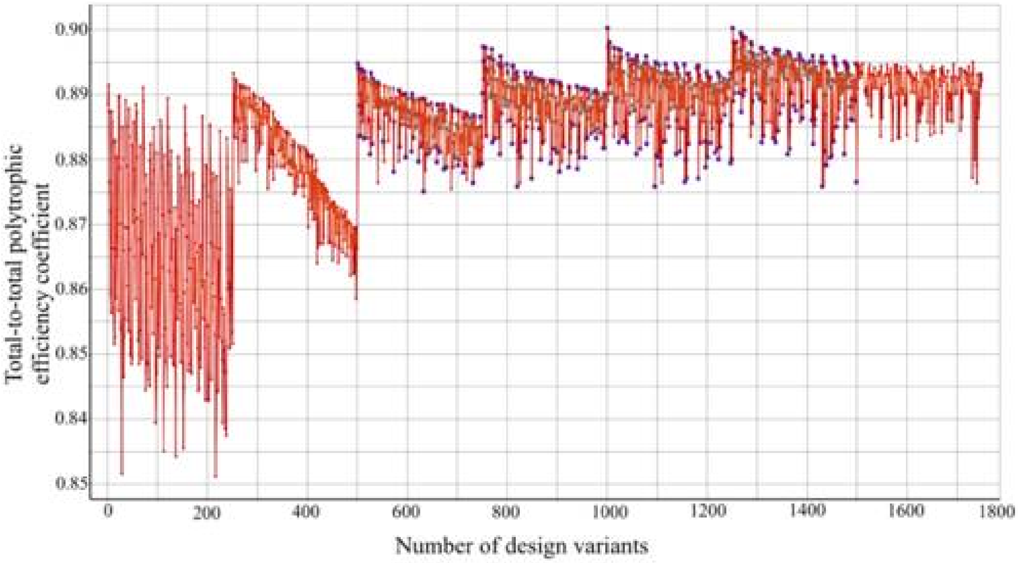

FIGURE 6. Results graphs of the efficiency versus calculation variants of PK-61 CFD model made by multi-objective optimization

The graph shows the process of carrying out optimization, initially the distribution of options for filling the ranges of variation of the selected parameters with obtaining results. Then, a response surface is constructed, along which the best optimization direction is determined. For the chosen direction, options are finally calculated. The chosen optimization variant is calculated separately for obtaining gas-dynamic characteristics, see fig. 7

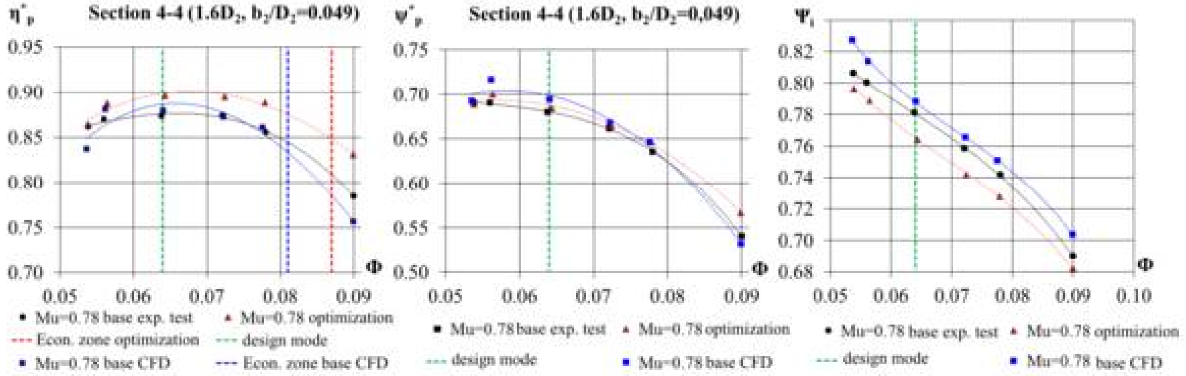

FIGURE 7. Scheme of the meridional and radial section of the impeller.

According to the results of optimization relative to the base case, the zone of economical work was increased by ~ 30%, the efficiency was increased by 1.6%. Consequently, this approach can be successfully used in fine-tuning the designed flow part to obtain the highest performance.

CONCLUSION

A complex of gas-dynamic design and profiling has been developed. In the process of theoretical calculation, a qualitative and quantitative assessment of the parameters characterizing the structure of the flow, and the losses in the elements, and the comparison of various variants of flow parts by efficiency is carried out. The problem is solved by calculating and analyzing the distribution of flow rates in the elements of the stage. Quantitative assessment of individual component losses and total losses is carried out according to the methods based on quasi-three-dimensional non-viscous and three-dimensional viscous flow models in the flow part.

In a general case, the complex allows the sequential execution of the following work steps:
1. Gas-dynamic calculation of the stage; profiling of stage elements on the basis of a rational distribution of speeds; calculation of losses in elements according to the mathematical model, approximate methods and viscous methods;
2. Preparation of the parameterized CFD-model and the implementation of automatic multi-criteria and multi-parameter optimization.
3. Final CFD calculation to confirm the declared gas-dynamic characteristics.

This approach allows you to initially determine the most optimal shape of the flow path with the required efficiency, and, if necessary, carry out optimization to achieve the best performance. In the future, it is expected to develop a model of simulation modeling of the gas-dynamic characteristics of a two-stage stage of centrifugal compressors based on the developed software package for theoretical and theoretical research.

REFERENCES

1. Simonov A. M. Research efficiency and optimal design of high-pressure centrifugal compressor stages. p.p. 164 – 188. Proceedings of the Scientific School of Compressor Engineering SPbSPU. Ed. prof. Yu.B. Galerkin.– Ed. SPbSPU, St. Petersburg., 2010 - 670 p.
2. Danilishin A M, Kozhukov Y V. Yun V K. 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. IOP Conference Series Materials Science and Engineering 08/2015; Volume 90(1):012047. DOI:10.1088/1757-899X/90/1/012046
3. C. H.Wu. “A general theory of three-dimensional flow in subsonic, and supersonic turbomachines of axial, radial and mixed-flow types”. Trans. ASME, Nov. 1952. pg. 1363-1380. 1952
4. K.P.Seleznev, Y.I. Biba, B.N. Savin, A.M.Simonov. Numerical prediction of turbulent flow in centrifugal compressor stage. Arch. Mech.,41,5, pp.735-746?, Warszawa, 1989.
5. Ahti Jaatinen-Värri, Aki Grönman, Teemu Turunen-Saaresti, and Jari Backman. Investigation of the Stage Performance and Flow Fields in a Centrifugal Compressor with a Vaneless Diffuser Hindawi Publishing Corporation International Journal of Rotating Machinery Volume 2014, Article ID 139153, 10 pages http://dx.doi.org/10.1155/2014/139153
6. Soo-Yong Cho, Kook-Young Ahn, Young-Duk Lee, and Young-Cheol Kim, “Optimal Design of a Centrifugal Compressor Impeller Using Evolutionary Algorithms,” Mathematical Problems in Engineering, vol. 2012, Article ID 752931, 22 pages, 2012. https://doi.org/10.1155/2012/752931.
7. Guo, Shuai & Duan, Fei & Tang, Hui & Chuan Lim, Seng & Sin Yip, Mee. (2014). Multi-objective optimization for centrifugal compressor of mini turbojet engine. Aerospace Science and Technology. 39. 10.1016/j.ast.2014.04.014.
8. Kang, Hyunsu & Kim, Youn-Jea. (2016). A Study on the Multi-Objective Optimization of Impeller for High-Power Centrifugal Compressor. International Journal of Fluid Machinery and Systems. 9. 143-149. 10.5293/IJFMS.2016.9.2.143.