How to cite / Как сослаться на статью:
Danilishin, A.M., Kozhukhov, Y.V., Neverov, V.V., Malev, K.G., Mironov, Y.R. The task of validation of gas-dynamic characteristics of a multistage centrifugal compressor for a natural gas booster compressor station (2017) AIP Conference Proceedings, 1876, № 020046. DOI:10.1063/1.4998866

The Task Of Validation Of Gas-Dynamic Of Characteristics Of A Multistage Centrifugal Compressor, For A Natural Gas Booster Compressor Station

A.M. Danilishin[1, a)], Y.V. Kozhukhov[1,b)], V.V. Neverov[1], K.G. Malev[2], Y.R. Mironov[2]

[1]Peter the Great St.Petersburg Polytechnic University)
[2] JSC «Kompressor komplex»

[a)]*Corresponding author: Danilishin_am@mail.ru*, [b)]*kozhukhov_yv@mail.ru*

Abstract. The aim of this work is the validation study for the numerical modeling of characteristics of a multistage centrifugal compressor for natural gas. In the research process was the analysis used misyachna interfaces and software systems. The result revealed discrepancies between the simulated and experimental characteristics and outlined the future work plan.

INTRODUCTION

After the opening for users of supercomputer center "Polytechnic" in 2016 at the Peter the Great Polytechnic university (ex. LPI them. M. I. Kalinina) at the Department of "Compressor, vacuum and refrigeration engineering" made possible the simulation of gas-dynamic characteristics (GDC) of a full-size multi-stage centrifugal compressors, by the methods of computational fluid dynamics (CFD). Due to the multistage setting of numerical simulation, there are additional costs in time, on the convergence, and increase the error of the calculation. The questions of carrying out of verification of CFD-calculations for the viscous flow in multi-stage centrifugal compressors arise. When calculating the viscous flow in a multi-stage flow part the simulation errors of the gas-dynamic parameters accumulate from element to element. Hence, the paper takes up questions of carrying out of verification CFD-calculations for the viscous flow on example of eight-stage centrifugal compressors. Discusses possible causes of inaccuracies, evaluated the qualitative and quantitative deviations from experimental values, the intended further steps of the study.

Modeling errors associated due to the complicated nature of gas flow in the stages of centrifugal compressors and therefore is sensitive to the choice and setting of semiempirical turbulence models, type of grid interface, choice of computational domains, accuracy of the computational grid, assigned initial and boundary conditions. Also need to add the complexity of the constructing such a flowpath. This is the appearance in the model of domains with input and output chambers, interdisc gaps and labyrinth seals. In most cases, it takes considerable time to build computational models than calculation of the blade rows calculation only.

Such modelling greatly using in industry for example by GE, RAND Company [1, 8] and other. Developing methods for automated multi-parameter and multi-criteria optimization [5]. Processes of verification widespread and successfully used for fine-tuning separate stages [7,9]. Unfortunately the problems of verification of multistage centrifugal compressors (more than 2 stages) is given little attention and work on it almost do not exist.

The aim of this work: verification study for the numerical modeling of characteristics of a multistage centrifugal compressor for natural gas for the full analysis model by the methods of computational fluid dynamics for perfect and real gas based on the comparison with the experimental data and development of recommendations on the calculation of this type of machine.

The object of study is eight-stage centrifugal compressor designed to work at booster compressor stations of natural gas. General view of the structure illustrated in (Fig. 1).

Department "of Compressor, vacuum and refrigeration engineering" in cooperation with JSC «Kompressor komplex» held the first phase of a verification study of numerical simulation of GDC eight-stage centrifugal compressor. The calculated characteristics were compared with the results of the air test on the open stand of JSC «Kompressor komplex» for an existing centrifugal compressor.

The calculations were performed in two software packages ANSYS CFX and NUMECA Fine turbo. Conducted validation and verification of the characteristics of the flow part. Was used a mesh with y+<1.

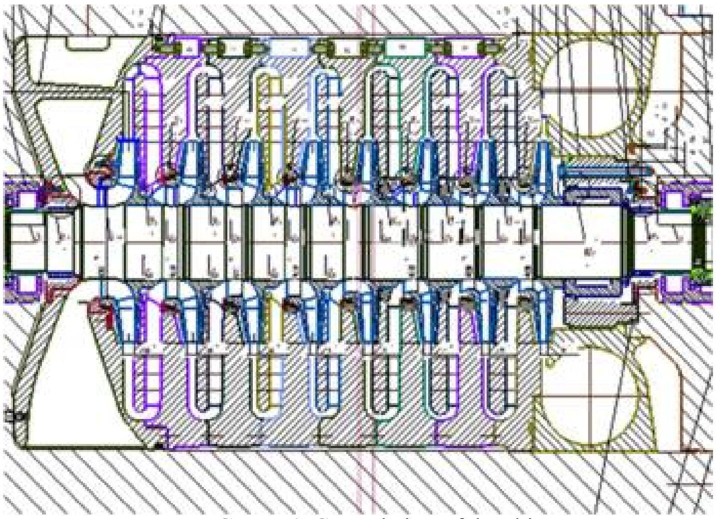


FIGURE 1. General view of the object

Field tests of the compressor were performed in the open air stand according to ISO 5389:2005 [3]. Measurement of flow parameters was performed in the sections "in-in", "out-out" in pipes with DN= 700 mm, where flow velocity <10 m/s. The tests were performed without insulation housing, the characteristics of the resulting parameters were recalculated taking into account heat transfer. General view of the stand depicted in figure 2.

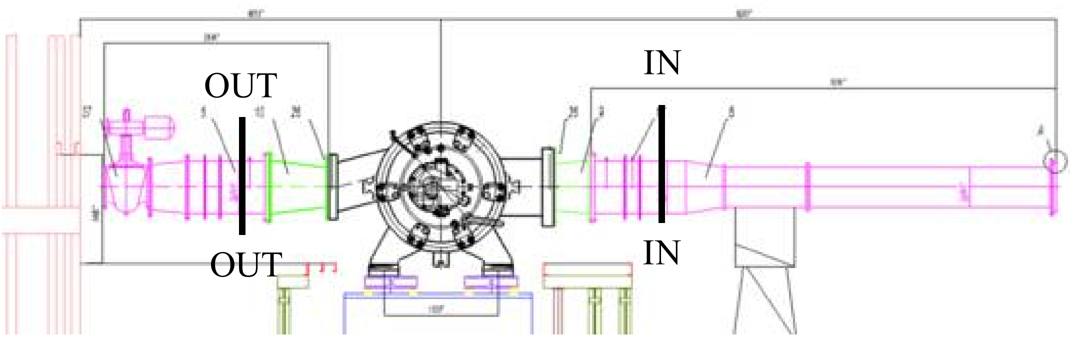


FIGURE 2. General view of the test stand

NUMERICAL CALCULATION

Department more than 20 years, step by step, conducts research for adaptation commercial CFD-package (TASCflow, Fluent, CFX, Fine/turbo) for a modeling flow parts of the centrifugal and axial compressors.

Initially conducted the studies on individual elements of a flow part: impellers, vane and vaneless diffusers, return blade channels. Noted about the real nature of the flow in the impeller of the centrifugal compressor and the possibility of flow analysis. In the studies was developed a methodology of the computation grid creation, setting boundary and initial conditions, solution, processing and analysis of the results. Then there was the modeling of the centrifugal compressor stages with a comparison with experimental data for integrated assessment and taking into account the modeling errors of the numerical calculation. It is concluded that for the investigated elements and stages, turbulence model SST (Shear stress transport) more accurately reflects the aerodynamic phenomena in the flow part in the design mode. Typical GDC presented by the polytropic efficiency, internal head coefficient, total pressure ratio. In all tasks at the design point the calculation error does not exceed ~4% [6]. Errors significant increases at the ends of the GDC - at off-design modes and up to more than 10%. The main reason of this mismatches are the overestimation of the internal head, characterized by the total enthalpy at the stage. For example, below figure 3 shows GDC for the stage with operational point at Φ_c=0,064.

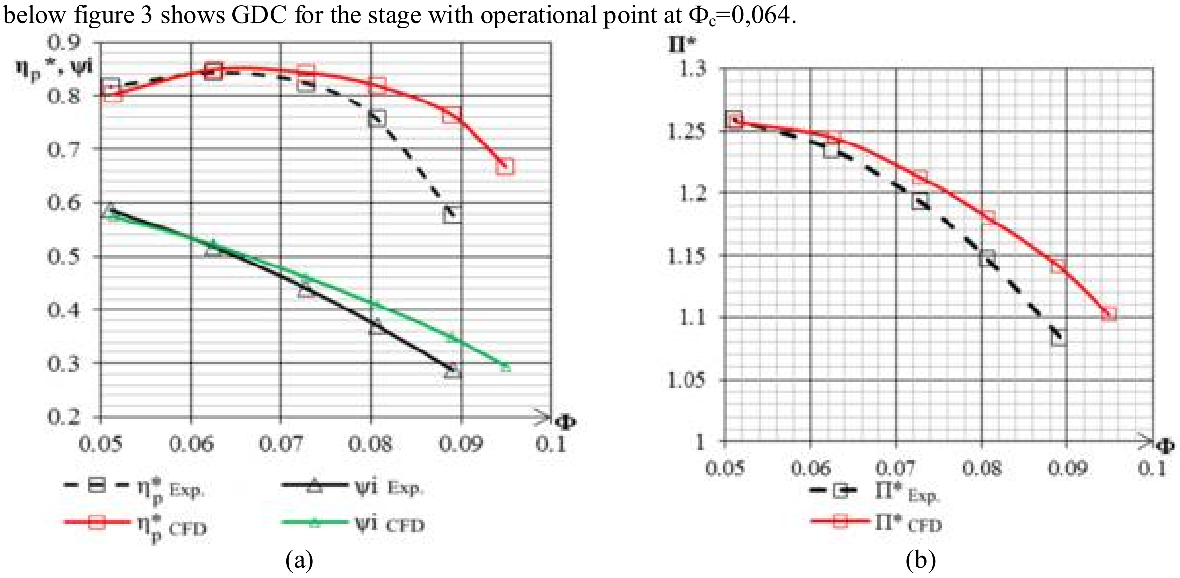


FIGURE 3. Experimental and CFD GDC of the centrifugal compressor stage with Φ_c=0.064, a) polytropic efficiency, internal head coefficient; b) total pressure ratio [6]

The question remains in the use of an grid interface: «Frozen rotor» and «Stage – mixing plane». From the physical point of view, the interface Frozen-rotor right simulated flow after the impeller: stream-wake, repeatedly observed experimentally. However, it gives an error in the calculation of the characteristics. Stage is more accurate in calculation, but averages the flow velocity over the circumference, including the turbulent Wake behind the edge of the impeller.

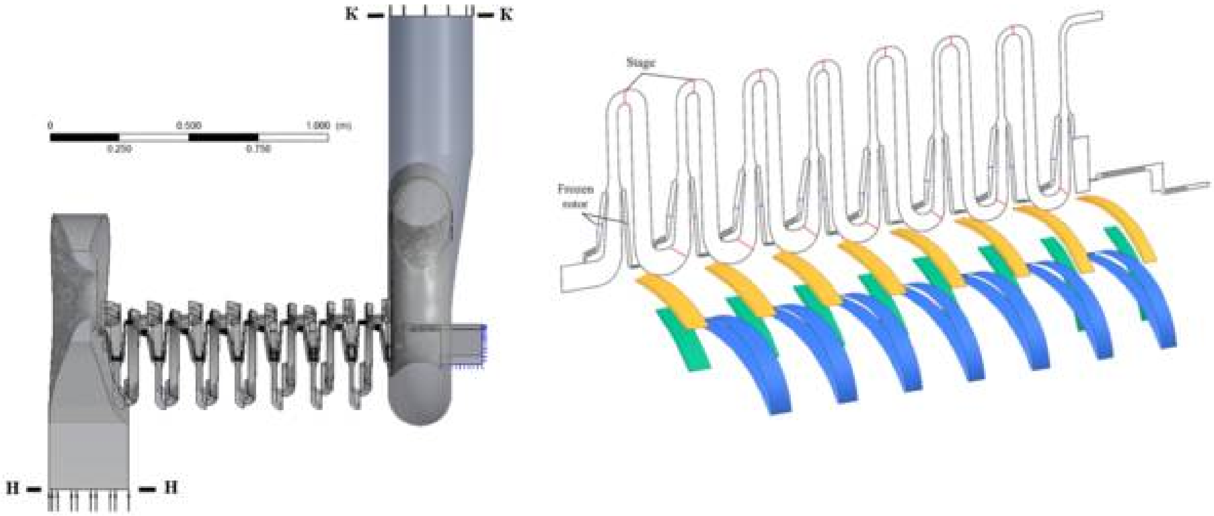
Apparently, for an adequate evaluation of performance in the future should abandon RANS-methods (stationary problem) and go to the solution of the URANS-methods (unsteady problem).

There is currently no General recommendation for the choice of interfaces, therefore in this work will be explored both interfaces to assess the quality of the calculation.

Originally was built the full model (Fig. 4 a) consisting of 8 steps, inter disc gaps and labyrinth seals, input and output chambers. Computational mesh for the full model was built for the blade rows in the program Autogrid5, and for diffusers, gaps, seals and chamber in ICEM CFD. There was a significant number of grid interfaces. Due to problems with convergence and the length of the calculation model has been simplified. The new model is made without the inlet and outlet chambers, i.e., a incomplete model built (Fig. 4 b). Computational mesh for incomplete model has fewer interfaces and was fully implemented in the program Autogrid5. General view of the computational mesh is presented in figure 6 and 7.

A dimensionless wall coordinate is calculated at y+=1 for use Low-Re turbulence model k-ω and Spalart-Allsmares. For all tasks was calculated stationary problem based on the RANS approach.

The calculations were performed in two computational fluid dynamics programs with the same mesh and boundary conditions, Ansys CFX v16.2 and Numeca FineTurbo v11.1. The calculation was done for five modes of performance to a fully converging the solution. Control of convergence was carried out according to the efficiency coefficient, calculated according to the input and output boundary. Additional control was carried out under the condition of conservativeness of the basic balance equations: P-mass, -V,-U,-W,-Mom and H-energy, expressed by imbalance of input and output flows across the borders of the computational domain. For a correct calculation of the value of the unbalance must tends to 0, the task is made acceptable value of unbalance equal to 0.01 % per domain. Before the main calculation was appropriate calculations to perform the mesh independence of the solution. Calculations without connections in the study also caused by the impossibility of calculation in the program Fine Turbo, as well as the desire of the authors to compare the results of simulations, i.e., to validate the calculation.



(a) (b)

FIGURE 4. a) full and b) incomplete CFD model of the eight-stages centrifugal compressor stage

Mesh of the incomplete model has about 80 million elements. General view of the mesh is presented in figure 5 and 6. The calculation was carried out at the supercomputer center "Polytechnic" with heterogeneous cluster HPC Tornado using 16 nodes (448 cores). The parameters of one node of the cluster: RSC Tornado – 2 CPU with 14 cores (2xXeon E5-2697v3 2.6 GHz 64 GB RAM). In sum - 712 nodes (19936 cores).

The simulations were run with P*inl and T*inl in inlet boundary and mass flow $\dot{m}_{out}$ at outlet boundary. The parameters correspond to measured values of field tests. For rotating items asked the frequency of rotation of the rotor. Used the turbulence model k-ω for ANSYS CFX, Spallart-Allmares for Numeca Fine Turbo. The working fluid is a perfect gas. The walls were assumed as adiabatic.

Feature of this problem is that in Ansys each stage is divided into rotor and stator parts to minimize the influence of grid interfaces on the calculation. Rotary part consists of the impeller, together with vaneless diffuser. The walls of the vaneless diffuser is stopped by the function of (counter rotating wall). The stator part consists of return blade channel, labyrinth seals and the inter-disk gaps.

For problem Ansys (Stage) interfaces is selected as: «frozen rotor» in the middle of the inter disc gaps, «stage» at the entrance to the impeller and at the mid-section of the rotary knee. It turns out 16 «stage» and 16 «frozen rotor» interfaces. For problem Ansys (Frozen rotor) 32 «frozen rotor» interface. Similarly spaced mixing plane interfaces (analogue stage) in a Fine Turbo.

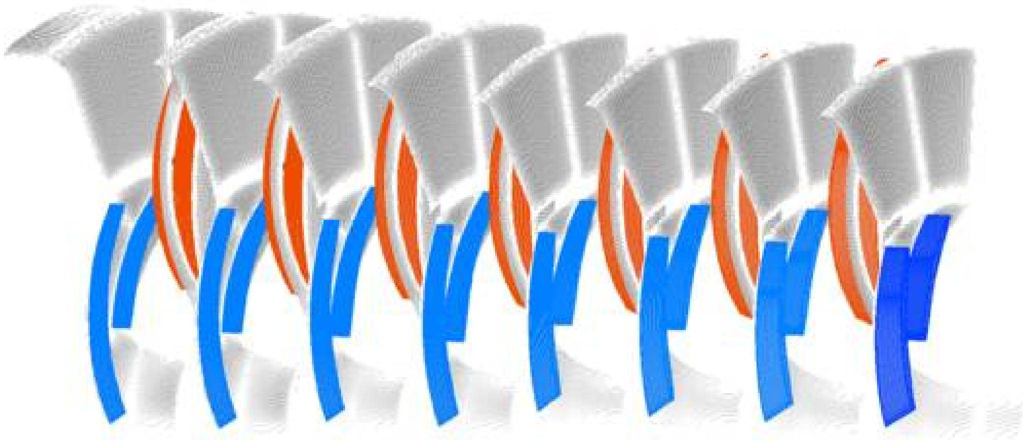
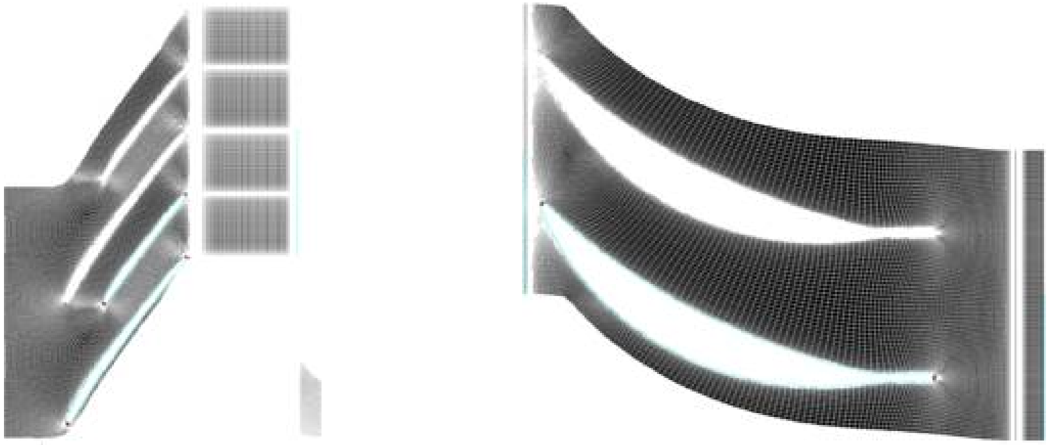


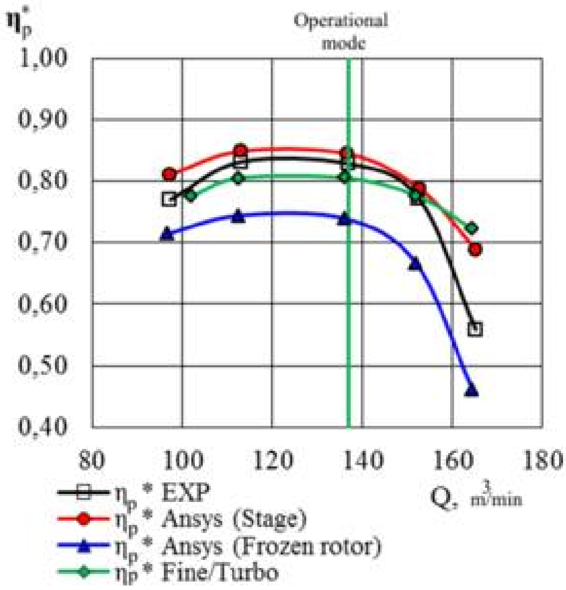
FIGURE 5. General view of mesh



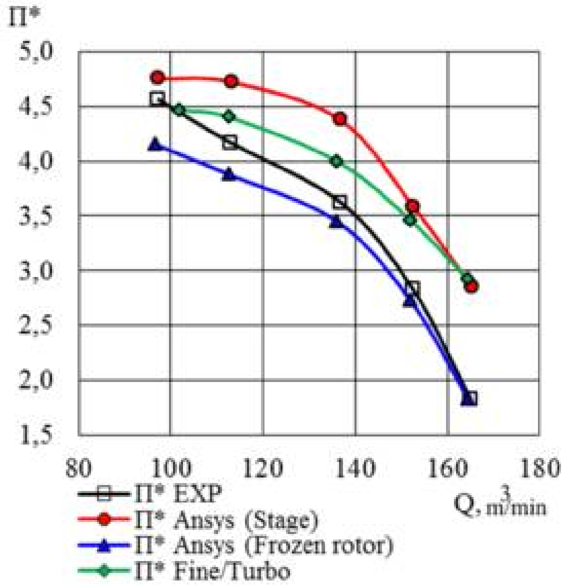
(a) (b)

FIGURE 6. View of the computational grid in a) impeller and vaneless diffuser b) return blade channel

Figure 7 shows the distribution of polytropic efficiency, pressure ratio in the compressor in dependence on the volumetric flow for the CFX and the Fine Turbo calculations with a comparison with the experimental data.



(a)



(b)

FIGURE 7. Graphs of the distribution of polytropic efficiency, pressure ratio, depending on the volumetric flow.

The results obtained in the calculation of perfect gas in CFX stage shows in the design mode is overstating the efficiency by ~1.2% and pressure ratio P* by 20.9% in relative units and ~0.01, 0,756 in absolute units.

When calculating Fine turbo lowering an efficiency of ~2.8% and pressure ratio, P* is 10.4% in relative units and by ~2.3% 0,376 in absolute units.

In the CFX calculation frozen rotor lowering efficiency at ~11.5% and the ratio of pressures P* 4.7% in relative units and ~9.6% and and 0.172 in absolute units.

The remaining modes are summarized in tables 1 and 2. Calculation of relative and absolute errors was carried out according to the following formulas:

$$\delta_i = \frac{\gamma_i - \gamma_{iexp}}{\gamma_{iexp}} \text{ and } \Delta = \gamma_i - \gamma_{iexp}$$

TABLE 1. The relative δ and absolute Δ polytropic efficiency error for the three calculations

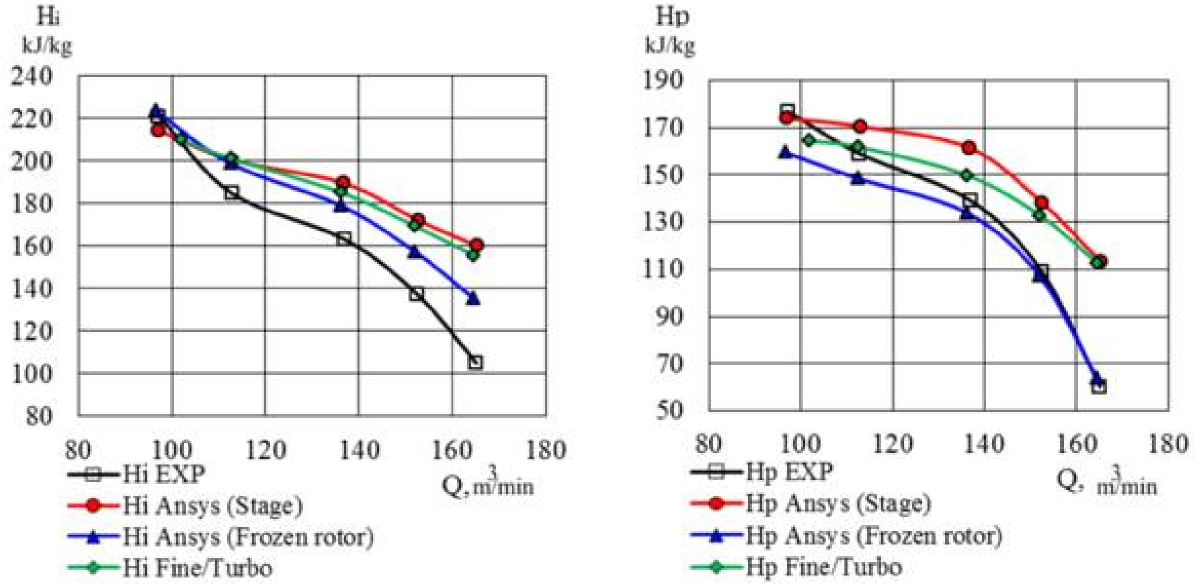
№	η_p* Fine/Turbo		η_p* Ansys (Stage)		η_p* Ansys (Frozen rotor)	
	δ,%	Δ	δ,%	Δ	δ,%	Δ
1	29,2%	0,163	23,2%	0,130	-17,6%	-0,099
2	0,6%	0,004	1,7%	0,013	-14,2%	-0,109
3	-2,8%	-0,023	1,2%	0,010	-11,5%	-0,096
4	-3,2%	-0,026	0,8%	0,007	-11,8%	-0,098
5	0,6%	0,005	3,5%	0,027	-9,0%	-0,069

TABLE 2. The relative δ and absolute Δ pressure ratio error for the three calculations

№	П* Fine/Turbo		П* Ansys (Stage)		П* Ansys (Frozen rotor)	
	δ,%	Δ	δ,%	Δ	δ,%	Δ
1	29,2%	0,163	23,2%	0,130	-17,6%	-0,099
2	0,6%	0,004	1,7%	0,013	-14,2%	-0,109
3	-2,8%	-0,023	1,2%	0,010	-11,5%	-0,096

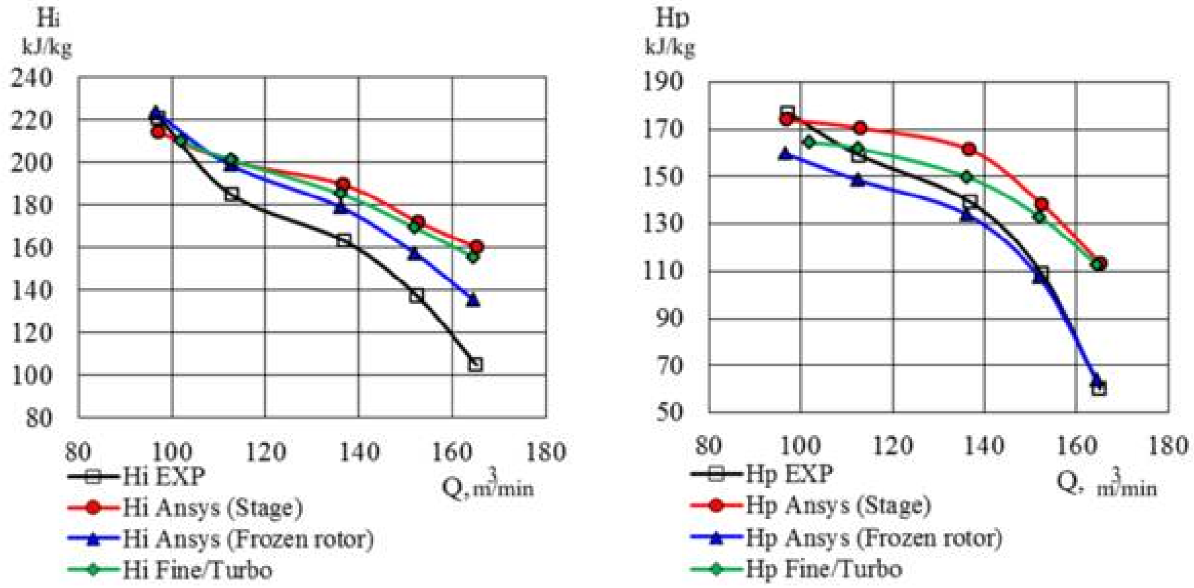
4	-3,2%	-0,026	0,8%	0,007	-11,8%	-0,098
5	0,6%	0,005	3,5%	0,027	-9,0%	-0,069

Figure 8 shows graphs of the distribution of internal and polytropic heads, figure 9 graphs of the distribution of pressure and internal power.



(a) (b)

FIGURE 8. Graphs of the distribution of internal and polytropic heads depending on the volume flow.



(a) (b)

FIGURE 9. Graphs of the distribution of outlet pressure and inner power depending on the volumetric flow.

Note that the calculation is performed without the input and output cameras that can significantly affect performance. In practice, the compressor effect both cameras reduces the efficiency of the entire machine by 4-8%. You also need to keep in mind the error in the tests is measured according to standard ±2%, and the fact that at the stage inlet pressure was set to be read in the section n-N in front of the suction chamber.

Given the above, closest to the experiment on the efficiency gave the calculation of Ansys (Stage) with the error on the current mode of 1.2%, the closest relative pressures was the calculation of Ansys (Frozen rotor) with an error in the design mode -4,7%. Quality noted the repetition of shape features Ansys (Frozen rotor) and testing. The discrepancy between the calculation of Numeca and CFX reach 4-9%, and is likely related to the used turbulence models. All calculations overestimate the internal head of the compressor.

CONCLUSIONS

Existing methods of numerical calculation for individual stages in a multistage centrifugal compressor quantitatively cannot be considered fully satisfactory. It is expected that with a proper diagnosis of the source of error is the possibility to compare the results of two calculations and obtain quite satisfactory for engineering problems result. To confirm this conclusion will be undergoing additional verification studies and testing methods for multistage compressors.

Quantitative differences when using a Stage or Mixing-plane is likely to be associated with the accumulated error of the averaging parameters for the impeller and their impact on the following elements after the UI stream mixes instantly and has a uniform around the circumference of the profile, which increases the effectiveness of items, so you should check the calculation for the individual steps of the task parameters of the output stage to the input of the next. This will reduce the time of computation and accuracy of numerical calculation, since it is known that the modeling of the characteristics of low-head and middle consumption centrifugal compressor stages qualitatively and quantitatively the same efficiency and pressure with the full-scale experiment within the bounds of acceptable in engineering calculations of error of ±2% in the zone of optimum performance [6]. Quantitative differences when using the Frozen rotor great for assessing efficiency, but with sufficient accuracy to correct for the relationship pressure.

Calculations of multi-stage machines in the RANS formulation can be used to estimate the ratio of the pressures in the compressor when using the Frozen rotor interface, the average error of the ratio of pressures is about 5% down on the current mode. They can also be used to determine the shape characteristics, as the characteristics qualitatively agree.

There is a need in the transient calculation of the characteristics of such calculations is recommended in a full ring setting, which causes an increased need for computing resources. However, there are methods that greatly accelerates the non-stationary calculation method, for example, nonlinear harmonics, which will be used later for Fine Turbo [4, 10]. There would also be non-stationary calculation in CFX. These calculations will allow to judge the applicability of computational fluid dynamics for the calculation of multi-stage machines and solving the problems of accuracy arising from the use of a large number of interfaces when merging the elements of the flow of in a single settlement area.

REFERENCES

1. C. Aalburg, A. Simpson, M. B. Schmitz V. Michelassi, S. Evangelisti, E. Belardini, V. Ballarini. Design and Testing of Multistage Centrifugal Compressors With Small Diffusion Ratios. Journal of Turbomachinery. JULY 2012, Vol. 134 / 041019-7DOI: 10.1115/1.4003715
2. ANSYS Inc. ANSYS CFX 16.2: Users Manual, 2016
3. ISO 5389:2005 Turbocompressors - Performance test code
4. Boldyrev, Y., Rubtsov, A., Kozhukhov, Y., Lebedev, A., Cheglakov, I., Danilishin, A. Simulation of unsteady processes in turbomachines based on nonlinear harmonic NLH-method with the use of supercomputers // CEUR Workshop Proceedings. Volume 1482, 2015, Pages 273-279.

5. Danilishin A., Kozhukhov Y., Yun V, 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
6. Danilishin A., Kozhukhov Y., Gileva L., Lebedev A. Verification of the CFD calculation on a supercomputer of medium flow model stages of centrifugal compressor. Russian Supercomputing Days: Proceedings of the international conference (September 26-27, 2016, Moscow, Russia). Moscow State University, 2016.
7. M. Elfert, A. Weber, D. Wittrock, A. Peters, C. Voss, E. Nicke. Experimental and numerical verification of an optimization of a fast rotating high performance radial compressor impeller. Journal of Turbomachinery. March 30, 2017. doi:10.1115/1.4036357;
8. James M. Sorokes, Jorge E. Pacheco, and Kalyan C. Malnedi. Pushing the Envelope. Ansys advantage. Volume VII | Issue 3 | 2013 pp.16-20
9. Klemens Vogel, Reza S. Abhari, Armin Zemp. Experimental and Numerical Investigation of the Unsteady Flow Field in a Vaned Diffuser of a High-Speed Centrifugal Compressor. Journal of Turbomachinery. JULY 2015, Vol. 137 / 071008-1/doi: 10.1115/1.4029175
10. Vilmin, S., Lorrain E., and Hirsch, C.: Unsteady Flow Modeling Across the Rotor/Stator Interface Using the Nonlinear Harmonic Method, ASME Paper GT-2006-90210, (2006)