Numerical Investigation of Different Radial Inlet Forms for Centrifugal Compressor and Influence of the Deflectors Number by Means of Computational Fluid Dynamics Methods with Computational Model Validation

Y V Kozhukhov[1], V K Yun[2], L V Reshetnikova[3], M V Prokopovich[4]

[1] Deputy Head of the Compressor Department, Vacuum and Refrigeration Engineering. Saint-Petersburg Polytechnical University, Russia, Saint-Petersburg

[2] Professor of the Compressor Department, Vacuum and Refrigeration Engineering. Saint-Petersburg Polytechnical University, Russia, Saint-Petersburg

[3] Student of the Compressor Department, Vacuum and Refrigeration Engineering. Saint-Petersburg Polytechnical University, Russia, Saint-Petersburg

[4] Engineer of the Design department of special engineering, Russia, Saint-Petersburg

Email: lyubov5reshet@gmail.com

Abstract: The goal of this work is numerical experiments for five different types of the centrifugal compressor's inlet chambers with the help of CFD-methods and comparison of the computational results with the results of the real experiment which was held in the Nevskiy Lenin Plant in Saint-Petersburg. In the context of one of the chambers the influence of deflectors on its characteristics was investigated.

The objects of investigation are 5 inlet chambers of different types which differ from each other by deflectors' existence and by its number. The comparative analyze of the results of numerical and real experiments was held by means of comparison of relative velocity and static pressure coefficient distribution on hub and shroud region, and also by means of loss coefficient values change for all five chambers.

As a result of the numerical calculation the quantitative and qualitative departure of CFD-calculations results and real experiment were found out.

The investigation of the influence of the number of deflectors on flow parameters was carried out. The results of the study prove that the presence of the deflectors on flow path significantly increases the probability of the flow separations and reversed flows appearance on them. At the same time, the complete absence of the deflectors in the chamber significantly increases circumferential distortion of the flow; however the loss coefficient decreases anyway, the high values of which are caused by the shock flow existence. Thus, the profiling of the deflectors of the inlet chamber should be given a special attention.

1. Introduction

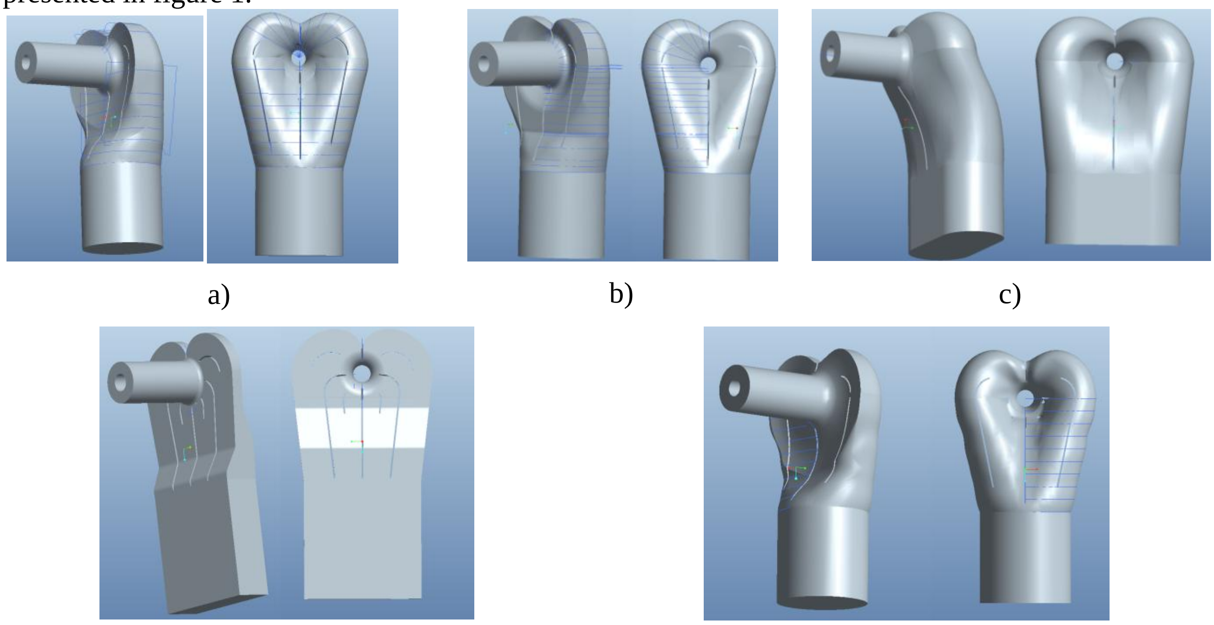
The radial inlet chamber is intended for a gas supply to the impeller of the first stage of the centrifugal compressor with the smallest energy loss and also a uniform flow profile into the inlet. From the quality of design, production and installation of the inlet chamber and, therefore, from aerodynamic perfection of its channels depends the value of the power consumed by the compressor on the commission of the work of gas compression to the demanded value of discharge pressure (P_{dis}). Wrong parametric choice of inlet device can become the reason for a decrease in efficiency of the centrifugal compressor of 3 … 5% and can considerable narrow down the range of its steady work. That is an essential negative factor, especially for compressors of high power [1].

Research into inlet devices made by the authors by means of CFD (Computational fluid dynamics) methods show that calculation in CFD programs are coordinated with the data of natural experiment. Both calculation and experiment confirm that changes of the design which will lead to the improvement of inlet device operation are possible. Based on this work and on the previous CFD calculations, the authors claim that similar programs can be used with confidence for the analysis, improvement and change of the radial inlet chambers. The unsteady flow phenomena which are observed at practical experiment show that CFD doesn't yield the full result at stationary problem definition RANS (Reynolds-averaged Navier–Stokes equations).

Also research shows that besides the traditional programs for a grid creation, such as ICEM CFD, Turbogrid, Solid Works, Fluent, the new programs can also be used. For example University of Florence uses Centaur Software and also TrueGrid for the hybrid grids creation of the radial inlet chambers [2].

Practical experiments for the inlet devices used in this work were held at the Nevsky plant as a part of a first model stage of a K4250-41-1 compressor in a single-cylinder performance [3]. This research was one of the sections of the design and research cycle of development works of the new type of K4250-41-1 compressor. This compressor is intended for the blasting pressure increase in the blast furnaces. The purpose of the practical experiment was clarification of influence of a form of the inlet chamber on the centrifugal compressor operation. Practical experiments were held for five various forms of the inlet chambers, including the option with axial intake, for the purpose of choice of the best variant.

Objects of this research in CFD programs are 5 computer models of the radial inlet chambers completely identical to the chambers tested at the Nevsky plant, further called RIC1, RIC2, RIC3, RIC4 and RIC5. CAD-models (Computer-aided design models) of investigated chambers are presented in figure 1.



a) b) c)

d) e)

Figure 1. CAD-models of the investigated chambers a) RIC1, b) RIC2, c) RIC3, d) RIC4, e) RIC5

The purposes of the investigation are:

1) Performance of calculations and analysis of the flow at five inlet chambers of the centrifugal compressor by means of the software package ANSYS CFX, and also the structure definition of the flow pictures on various parts of computational space;

2) Calculation of the relative velocity $\overline{c}_2$, the static pressure coefficient K_P and the loss coefficient ζ for each chamber;

3) Comparison of received results of the modeling with the Nevsky plant experimental research data and conclusion about the possibility of the software package ANSYS CFX application for the flow calculation in the inlet chambers of centrifugal compressors;

4) Researches of the deflectors influence on one of the inlet chambers and the conclusions composition about the advisability or inadvisability of any changes introduction into the chamber design.

As a result of the carried-out work it is necessary to reveal an optimum construction of the inlet chamber among the investigated variants. It is also necessary to introduce the conclusion about the possibility of the software package ANSYS CFX application for the inlet chambers of the centrifugal compressors improvement.

2. Experimental investigations:

For carrying out experimental investigations of the single flow variant of the K4250-41-1 compressor the model of the first stage have been designed and manufactured. The model of the stage was made on a scale of $i = \frac{D_2}{D_2^*} = 5{,}41$ and was driven from the 125 kW electric motor, 380V, 2950 rpm through the increasing gearbox with transfer number of $i_p = 5{,}6$. The flow path of the model consisted of the impeller with the diameter of $D_2^* = 305\ mm$, the vaneless diffuser with return directing device and the outlet radial diffuser which imitated the flow entrance to the second stage. Air intake to the impeller was made from removable inlet chambers.

3. Analysis of the results of the experimental investigation:

Parameters on the inlet to the chamber: total temperature $T^* = 293\ K$, total pressure $P^* = 98100\ Pa$, mass flow $\overline{m}_p = 2{,}2$ kg/s, gas – air.

Based on the research the best characteristics were shown by the RIC1 chamber.

The RIC3 chamber which is characterized by the smoothest turns in the flow path is approximately equivalent to the RIC1 chamber. The flow inhomogeneity at an outlet from this chamber is slightly less, than behind the RIC1 chamber.

The next chamber by quality is the RIC5 chamber which has smooth turns in the flow path and is designed with higher acceleration of the flow in the middle part of the chamber than in the RIC1. The pressure loss ζ at this variant of the inlet chamber reaches 8,3% in comparison with an axial inlet. However, looking at the efficiency this chamber is close to the RIC1 and the RIC3 variants.

The RIC2 chamber is characterized by sharp turning of the chamber at the covering impeller disk. The model test of this chamber showed that loss of pressure increase in comparison with axial inlet on the design condition reaches 13,6%. Axial asymmetry of the flow and its inhomogenuity by radius reaches higher volume, than in the previous chambers.

The RIC4 chamber is small-sized, with the clamped sections and small curvature radiuses in flow path gives increase in pressure for 15% less at design mode, than at axial inlet and this chamber is the worst of all.

The loss coefficients ζ of all the studied chambers are compared on figure 2. By the loss coefficient the inlet chambers are distributed approximately in the same order, as on the basis of analysis of gasdynamic characteristics and uniformity of the flow at an outlet from the chambers. The RIC3 and RIC1 chambers have the smallest loss coefficient and RIC4 and RIC2 have the greatest. The loss coefficient for the chamber is determined by the formula: $\zeta = \frac{P_{1mid}^* - P_{2mid}^*}{\rho_{2mid} \cdot \frac{c_{2mid}^2}{2}}$

Characteristic feature of the investigated flow path of the stage is high $\frac{c_0}{u_2}$ value (ratio of the flow velocity before the impeller to the velocity on the impeller rim) which reached the value $\frac{c_0}{u_2} = 0{,}4$. This circumstance strengthened the role of the inlet chamber and its influence on the gasdynamic characteristics of the stage. The works which were carried out on the model with low ratio $\frac{c_0}{u_2} = 0{,}09$ showed that at such low $\frac{c_0}{u_2}$ values the role of the inlet chamber is much less, and it is possible to obtain gasdynamic characteristics of the stage with the inlet chamber installation, almost no different from characteristics with axial inlet.

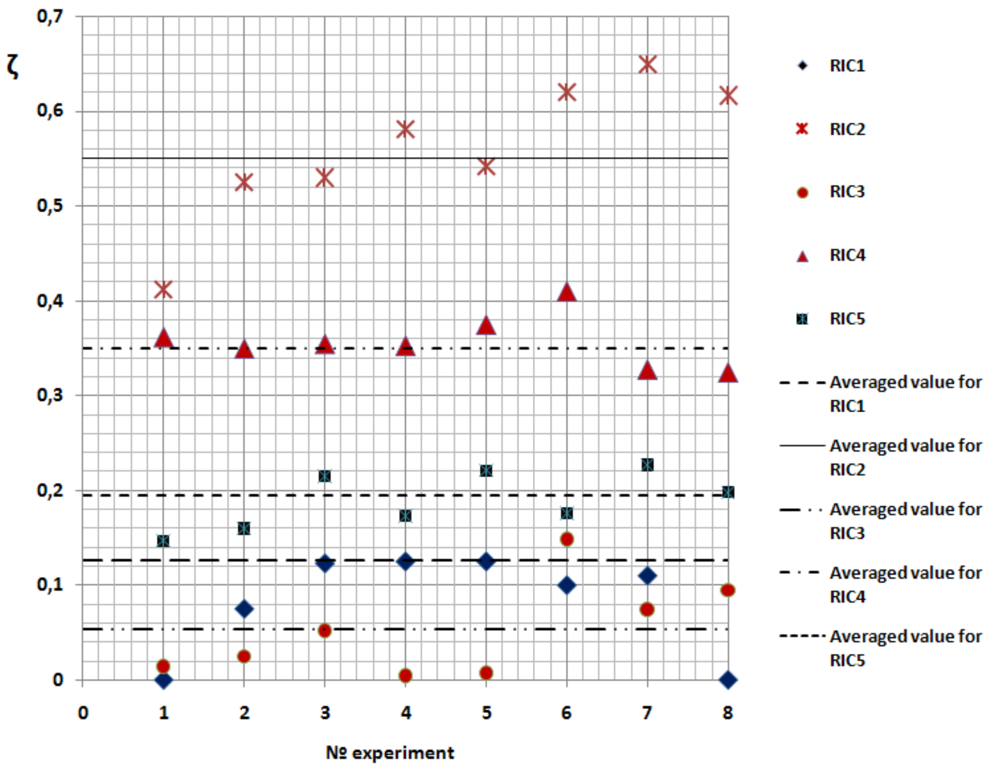


Figure 2. Experimental data of the loss coefficients of the investigated chambers with the values averaging

The following conclusions are drawn by results of the research:

1) The asymmetry of the flow in the output diffuser which distorted work of the stage took place. This occurred because the support plate was too close to the outlet confusor of the chamber.

2) Change of an operating mode of the stage by means of the replaceable diaphragms in the inlet, caused pressure decrease to 3000 mm w.c., which brought difficulties to the pressure measurement of the chambers.

3) Asymmetry of the outlet flow in the diffuser created an uneven temperature field that didn't allow correct estimation of the power consumption and the efficiency of the installation.

4) Measurement of total pressure heads at the outlet, from the return directing device, was performed only by one radius location.

5) Specified circumstances didn't allow research to the desired accuracy and detail. Nevertheless, the measured results allow a judgment to be made about the quality of the inlet chambers.

6) From five types of the inlet chambers which were exposed to these investigations, the best chamber by its qualities was the RIC1 chamber, the worst – the RIC2 chamber.

5. CFD- investigation of the inlet chambers of the centrifugal compressor:

During the numerical investigation 5 different models of the inlet chambers of the centrifugal compressor were under consideration. Their names are the same RIC1, RIC2, RIC3, RIC4, RIC5.

Preparation for the calculation of all radial inlet chambers was carried out in 3 steps:

1) Creation of the radial inlet chamber CAD-model in the program complex Pro/ENGINEER Wildfire 5.0. For creation of all 5 chambers the drawings were used which were provided by the Nevsky plant. The extended outlet section was created after the confusor for each chamber. This section was made for investigation of the flow unevenness which changes along its length after the confusor. CAD-models of investigated chambers are presented in figure 2.

2) Unstructured grid creation in the program ANSYS ICEM CFD 14.0 module with necessary concentration at walls. In the flow core the elements are tetrahedrons and 15 prismatic layers at walls.

3) In CFX-Pre module the boundary conditions for calculation were set. For obtaining the characteristics at different mass flows two types of boundary conditions were used:

- Total pressure and temperatures at the inlet and outlet of the chamber with a condition of two-way entry of the flow through the inlet and outlet (boundary condition “Opening”);
- Total pressure and temperature at the inlet and boundary condition “Inlet”, mass flow and boundary condition “Outlet” at the outlet.

Parameters at the inlet correspond to experimental conditions: total temperature $T^* = 293\ K$, total pressure $P^* = 98100\ Pa$, mass flow $\overline{m}_p = 2{,}2$ kg/s, gas – air.

Calculations were made with turbulence model SST as it is the most universal and is developed on the basis of the verified two-parametrical k-ε and k-ω models, uniting their best properties.

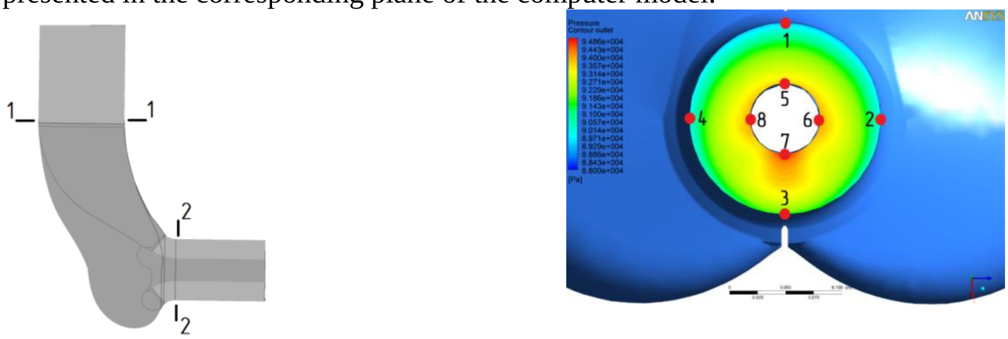
The mass flow $\overline{m}_p = 2{,}2\ kg/s$ is taken as the design mode of the investigated chamber which also corresponds to minimum loss coefficient at the chambers and to maximum efficiency of the stage. This mass flow also corresponds to design mass flow of the model chamber of the Nevsky plant. As during the experiment, all chambers were investigated at the test-stand with the same model stage, the design mass flow is identical at every chamber because the design mass flow of the stage doesn't change.

6. Analysis of the numerical experiment results:

As a result of numerical calculation when considering the loss coefficient value the best is the RIC3 chamber, the worst is RIC4. The others are intermediate between them.

The results analyses were carried out for each chamber. The maximum velocity appears on the peripheral surface of the confusor from the inlet when turning the flow. But, this chamber is characterized by smoother turns of the flow; therefore, there is less probability of the separation.

On figure 3 a) the placement of sections 1-1 and 2-2 is shown corresponding to the inlet and outlet sections of model chambers of the Nevsky plant. On figure 3 b) the scheme of static pressure tapping points in section 2 for model chambers of practical experiment and also a field of static pressure is presented in the corresponding plane of the computer model.



a) b)

Figure 3. Results of the CFD modeling for the RIC3 inlet device a) the placement of sections 1-1 and 2-2; b) the scheme of static pressure tapping points in section 2 for the model chamber of the Nevsky plant in the corresponding plane of the computer model; static pressure field

On Figure 4 the comparison of the relative velocity distribution $\bar{c}_2 = c_{2i}/c_{2mid}$ in section 2 is presented, where i – the static pressure tap point which was measured by holing the chamber walls.

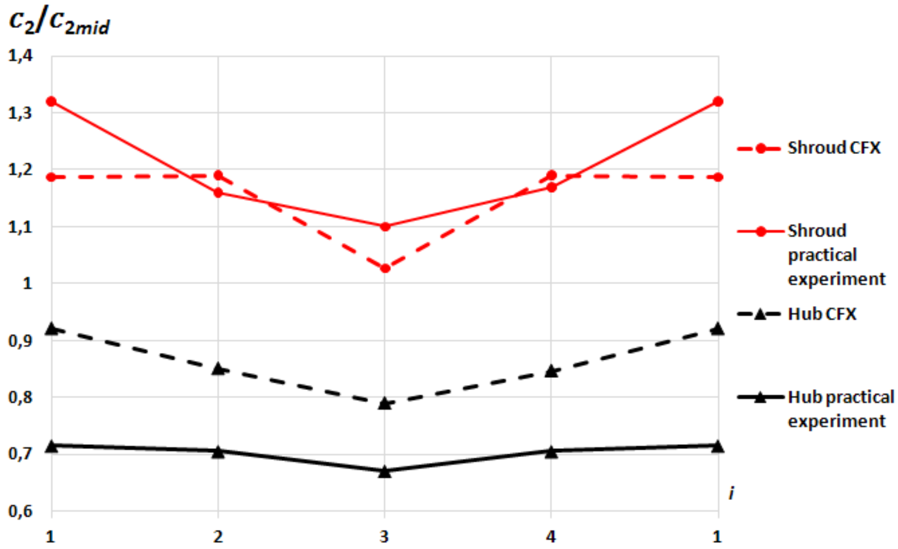


Figure 4. Comparison of the relative velocity distribution $\bar{c}_2 = c_{2i}/c_{2mid}$ in section 2 for the inlet device RIC3.

From figure 4 the qualitative coincidence of the velocity distribution is visible. The quantitative difference can be explained with the error of measurements at practical experiment and inaccuracy of iterative calculations by CFD methods.

On figure 5 the static pressure coefficient K_P distribution by circle in section 2-2 of the RIC1 inlet device is presented, where i – is a point of the static pressure tap. Loss coefficient is determined by the formula:

$$K_P = \frac{P_{1mid} - P_{2i}}{\rho_{2i} \cdot \frac{c_{2mid}^2}{2}}, \tag{1}$$

where P_{1mid}– static pressure on the inlet section in front of the inlet chamber, settling down behind the straight section of constant diameter; P_{2i} and ρ_{2i}–static pressure and density in section 2 in the point of pressure tap i respectively.

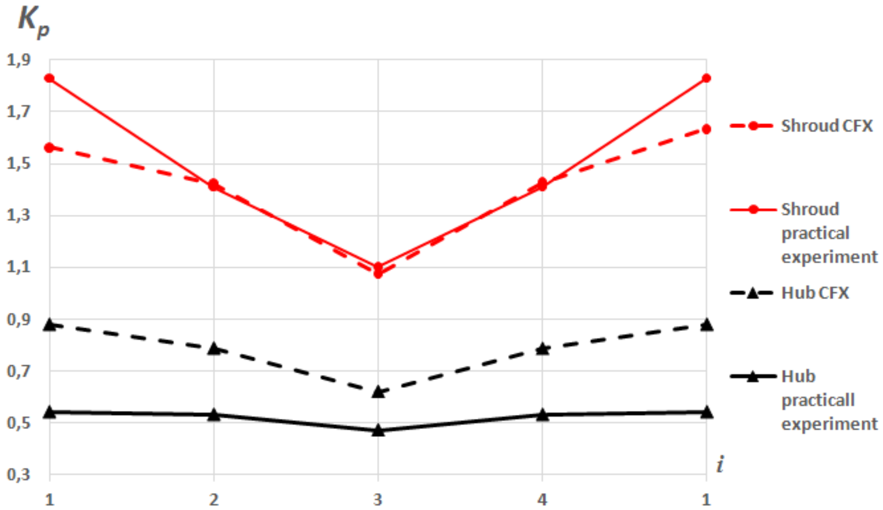


Figure 5. Comparison of the static pressure the K_P coefficient distribution by circle in section 2-2 of the RIC3 inlet device

The loss coefficient for the chamber is determined by the formula:

$$\zeta = \frac{P^*_{1mid} - P^*_{2mid}}{\rho_{2mid} \cdot \frac{c^2_{2mid}}{2}}, \tag{2}$$

where P^* - total pressure.

By the results of the practical experimental investigations at the Nevsky plant the loss coefficient of the chamber on the design mode equals $\zeta_{exp_p} = 0{,}05$.

The loss coefficient of the chamber on the design mode by the results of calculation in ANSYS CFD:

$$\zeta_{CFD_p} = \frac{P^*_{1mid} - P^*_{2mid}}{\rho_{2mid} \cdot \frac{c^2_{2mid}}{2}} = \frac{98096{,}7 - 97812{,}375}{1{,}117 \cdot \frac{96{,}2^2}{2}} = 0{,}055 \tag{3}$$

The difference of ζ calculation from the experimental value is about 9%, and $\zeta_{CFD_p} > \zeta_{exp_p}$. This error is admissible and shows good similarity of calculations for the inlet device.

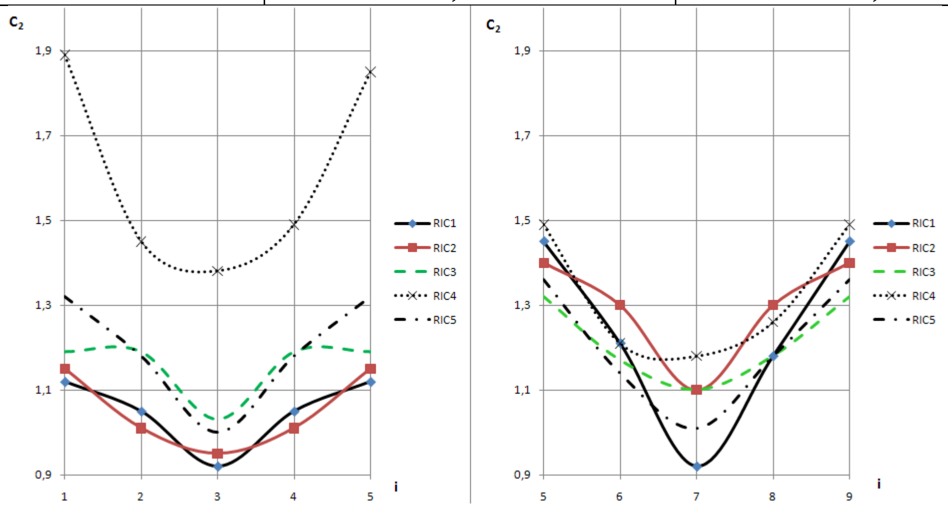
7. General conclusions by the results of calculations for five chambers:

From the results of calculation for all five chambers it is possible to make the general conclusions for the results of calculation and their similarity to the results of practical experiment of the Nevsky plant. It is also possible to make the comparative analysis of all five chambers by the results of numerical calculation.

For example on figure 6 the summary plots are shown which are received for the comparison of relative velocity $\bar{c}_2$distribution on the shroud area. Also such plots were received for the hub area for the relative velocity and the static pressure coefficient K_p on the hub and shroud area. Data of the loss coefficient values ζ are summarized in table 1.

Table 1. Loss coefficient ζ for five different chambers

Chamber variant	Loss coefficient ζ	
	CFD results	Experimental
RIC1	0,13	0,11
RIC2	0,115	0,55
RIC3	0,055	0,05
RIC4	0,293	0,35
RIC5	0,092	0,19



a) b)

Figure 6. The relative velocity distribution $\bar{c}_2$ on the shroud area for five chambers: a) by the results of the CFD calculation, b) by the results of the practical experiment on the Nevsky plant

From the table 1 it is visible that the RIC1, RIC3 and RIC4 chambers correspond to experiment results of the numerical calculation best of all. The result received at the practical experiment for the RIC2 is in doubt as in comparison with chambers of the similar configuration the value of the loss coefficient is unreasonably overestimated to value $\zeta = 0{,}55$. Various negative factors which occurred while carrying out the experiment may be the reason.

As it was already told, as a result of numerical calculation by the loss coefficient value the best chamber is the RIC3 chamber, the worst is the RIC4. The others are intermediate between them.

From figure 6 it is visible that the nature of the calculated relative velocity distribution corresponds to the received values of the practical experiment. Thus, qualitative similarity between the results of calculation and experiment is available. Quantitative similarity is not observed at all chambers. The greatest divergence with the practical experiment is shown by the RIC2 and RIC4 chambers. One of the reasons for inaccuracy could be that at practical experiment the values on the hub region could cause difficulties and that could distort the results.

Estimating smoothness of the flow in the chambers, it is possible to notice that the best characteristics are at the RIC1 and RIC2, there is no flow separation in them after turning. Contraflows at the small central deflector could be observed at the RIC1 and RIC5 chambers. Separations and contraflows are especially accurately investigated at the RIC1 and RIC5, but also present at the RIC2 and RIC4.

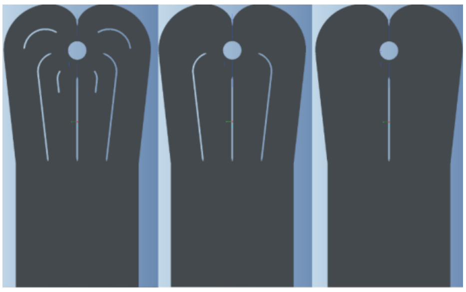
In general, the results received from numerical experiment don't contradict the results of practical experiment and reflect the flow picture in chambers. Thus similar calculations are recommended for research and optimization of the inlet chambers of centrifugal compressors.

8. Research of the deflectors influence on the chamber's efficiency:

For research of the deflectors influence on the chamber's efficiency the RIC4 chamber was chosen since its design allows making the changes more evident.

Having analyzed the results of the numerical experiment, it was revealed that the best characteristics the RIC3 chamber possesses in which there are no side deflectors. Therefore the decision was made in the course of investigations to take away at first the part, and then the others side deflectors out of the RIC4 chamber.

On figure 7 the distinctions in the geometrical form between initial chamber variant and two changed chambers of the RIC4 are presented. Further the changed RIC4 chamber variants without short deflectors and completely without them are named RIC4-v1 and RIC4-v2 respectively.



a) b) c)

Figure 7. CAD-models of the RIC4 chamber variants: a) RIC4, b) RIC4-v1, c) RIC4-v2

From the pattern of flow around the side deflectors which were received as a result of the calculations their shock flow was found out. The central deflector is flowed around in all cases without shocks.

For the changed chambers distributions of the relative velocity c_2/c_{2mid}, static pressure K_p and loss coefficient ζ were calculated. Values of loss coefficients calculated in CFD for three different variants are summarized in table 2. Values of relative velocity calculated in CFD and static pressure coefficient in comparison with the same sizes for an initial chamber RIC4 are given in figure 8 and figure 9

IOP Conf. Series: Materials Science and Engineering **90** (2015) 012047 doi:10.1088/1757-899X/90/1/012047

Table 2. Loss coefficient ζ for the RIC4, RIC4-v1 and RIC4-v2

Parameters	Chamber variants		
	RIC4	RIC4-v1	RIC4-v2
Loss coefficient ζ	0,293	0,291	0,229

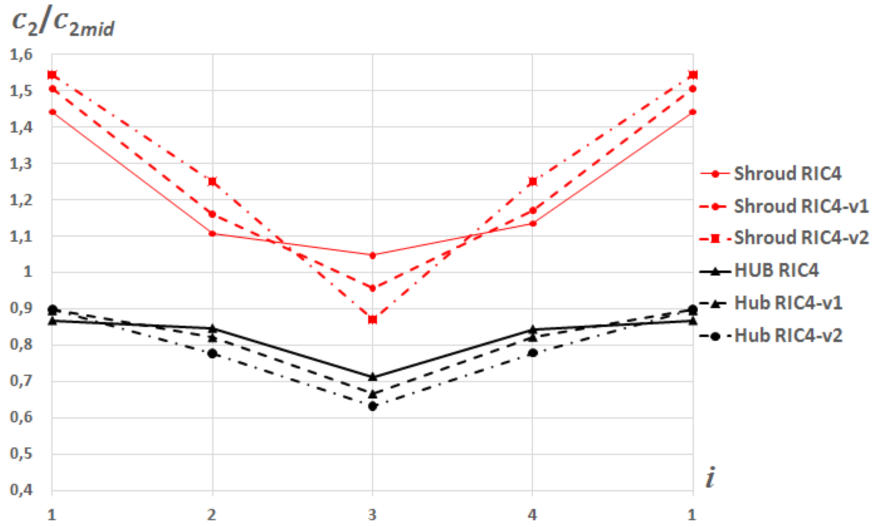


Figure 8. Comparison of the velocity distribution $\bar{c}_2 = c_{2i}/c_{2mid}$by the results of the CFX calculation.

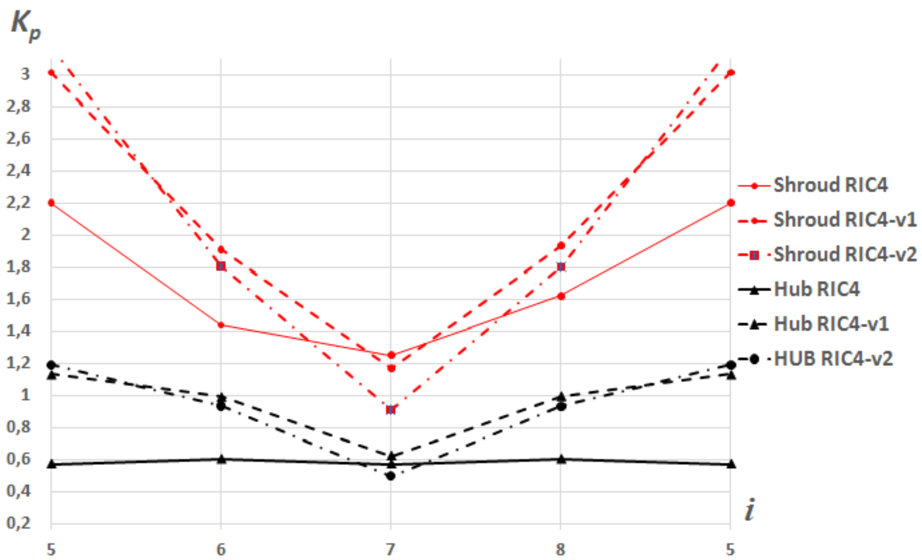


Figure 9. Comparison of the static pressure coefficient K_P distribution by the results of the CFX calculation.

The result when part of the deflectors was taken away from the RIC4 chamber, the loss coefficient did not change significantly while the flow became less uniform. Unevenness is explained by dividing deflectors being taken away from the chamber.

The chamber completely without deflectors shows better value of the loss coefficient due to the lack of loss arising at the flow while flowing around the deflectors. Unevenness of the flow at the chamber without deflectors has the greatest value in comparison with the variant with deflectors. Thus existence of the dividing deflectors in the chamber brings bigger loss, but does make the flow more uniform. Therefore, when designing the inlet chambers it is needed to pay special attention to the existence of the deflectors.

9. Conclusion:

The purpose of this work was to investigate five variants of the inlet radial chambers of the centrifugal compressor develop and research of the deflectors influence on the efficiency of one of the chambers tested at the Nevsky plant.

Received results were compared with the data of experimental investigations of the Nevsky plant by three parameters: relative velocity distributions $^{c_2}/_{c_{2mid}}$, static pressure coefficient $K_p$at the outlet section of the chamber and the loss coefficient ζ of the chamber. Research of influence of the side deflectors on parameters of the chamber's flow is made. The results of research prove that the presence of deflectors in the flow increases the probability of the flow separation and reverse flows on them. Full lack of deflectors in the chamber considerably increases the circular unevenness of the flow, thus nevertheless the loss coefficient from 0,293 to 0,229 where high values are caused by existence of the shock flow of deflectors. It is also possible to note the possibility of the analysis of the flow visualization when calculating in ANSYS CFD which allows defining the regions of the flow separation and contraflow in the chamber.

Results of the calculation show good compliance to the real flow and can be used for further researches and recommendations to use the ANSYS CFD for calculations of inlet radial chambers of centrifugal compressors. The possibility to increase the efficiency of inlet devices of turbocompressors is shown by the CFD methods.

At the same time it is necessary to specify that definition of pressure loss for the whole flow path of the centrifugal compressor or the stage has to be made for the whole compressor in general. It is connected with the fact that different elements of centrifugal compressor such as inlet chamber, impeller, diffuser etc. influence on each other in the flow. This fact has been confirmed in the practical experiment for the inlet chambers as a part of the compressor stage described in this work.

References:

[1] N A Karnauhova, Y V Gakerkin, Y V Kozhukhov. Research of the three dimentional flow at the radial inlet device of the centrifugal compressor with the use of the software package of ANSYS CFD, *the XL Week of science SPbSPU: materials of the international scientific and practical conference, SPb, Publishing house of Polytechnical university*, 2011.
[2] "Issues Surrounding Multiple Frames of Reference Models for Turbo Compressor Applications", *Z.Liu, D.L Hill, International Compressor Engineering Conference*, 2000.
[3] V K Yun. The analysis and assessment of the design methods of the flow path of centrifugal compressor machines//*Works of the XV international scientific and technical conference on compressor equipment. Part I. Kazan, 2011* pages 362-370.