PAPER ID: 0430

DOI: 10.18462/iir.compr.2020.0430

DESIGN PRINCIPLES FOR CENTRIFUGAL COMPRESSOR DIFFUSERS BASED ON THE USE OF VELOCITY DISTRIBUTION

Mykola Kalinkevych[a], Andrii Lazarenko[a], Mykola Radchenko[b]

[a] Sumy State University
Sumy, 40007, Ukraine, vikola58@ukr.net
[b] Admiral Makarov National University of Shipbuilding
Mykolaiv, 54000, Ukraine, nirad50@gmail.com

ABSTRACT

This paper describes the design principles of diffusers based on physical and mathematical models of the flow of swirling viscous compressed gas. According to the presented method, the diffusers' design is based on the preseparation condition of the boundary layer along one of the outer surfaces. Numerical and experimental research was performed for different compressor stages. Conducted experimental calculation studies and comparison of design and experimental characteristics of diffusers and flow parameters in diffusers confirm the efficiency of the diffuser, which was designed using the presented method and the adequacy of mathematical models with the physical processes of gas flow in diffusers. A new method of designing the vane diffusers and channel diffusers with a predetermined velocity distribution on vane surfaces provides an improvement in the gas-dynamic characteristics of diffusers compared to traditional geometry diffusers.

Keywords: Centrifugal Compressor, Method of the Design, Vane Diffuser, Channel Diffuser, Research

INTRODUCTION

Centrifugal compressors are widely used in many industries, including the refrigeration industry. One way to reduce the energy consumed by the compressor during its operation is to increase the efficiency of the compressors. Compressors efficiency improvement can be achieved by the improvement of the gas-dynamic characteristics of the elements of the compressor stages, including diffusers.

Much of the energy (40–60%) transmitted by the gas in the impeller of the centrifugal compressor is kinetic. Diffusers are used to convert the kinetic energy of a gas stream into a potential one, that is, to reduce the velocity and increase the pressure. The most common types of diffusers are vaneless, vane and channel.

The traditional design of diffusers of centrifugal compressors can be characterized as geometric, in which simple surfaces and lines are used – a plane, a straight line, an arc of a circle, etc. So, for vane diffusers, the middle line of the blades is defined by an arc of a circle. With such a design, a separation of the flow on the pressure surfaces of the vane is typical, while the separation point is located close to the inlet edge ($\bar{l}_{sep} = 0.2 - 0.3$). For channel diffusers, when constructing channels with straight walls, the recommendations are used, which are obtained by studying the direct-axis gas flow in diffuser channels. The gas flow after the impeller is swirling, and therefore the use of such recommendations for swirling gas flow is fundamentally wrong. Obviously, the creation of new design methods for vane and channel diffusers based on the use of gas-dynamic gas parameters, in particular the velocity distribution, while solving the inverse gas dynamics problem, provides the production of diffusers having better gas-dynamic characteristics, which improves the performance of the compressors.

For developing methods for designing vane and channel diffusers, the ideas of Stratford (1959) were used, according to which a separation criterion could be used to choose the form of the surface of the flowing body in such a way that the spring load of the surface rub in the current flow for an entire hour would be close to zero. Under this condition, the current at the edge of separation, still remains attached to the surface of the body. This allows for not only minimal resistance but also very low heat flux, since convective heat exchange is associated with surface friction. At the point of flow separation, the maximum pressure gradient is reached. Therefore, for a given boundary layer, the zero-friction condition allows to obtain a given allowable pressure increase at the shortest section and with the least energy dissipation.

THE MATHEMATICAL MODEL

The swirling flow of compressible viscous gas is considered. A mathematical model for solving the inverse gas dynamics problem is formed for a steady, unbroken flow of gas.

The angular momentum changes from the diffuser inlet to the current values about the axis for the annular element of gas:

$$\Delta M = \dot{m} \cdot \left(r_{in} \cdot C_{in} \cdot \cos\alpha_{in} - r \cdot C \cdot \cos\alpha \right), \qquad \text{Eq. (1)}$$

where $\dot{m}$ is mass flow; $r_{in}, C_{in}, \alpha_{in}$ is radius, velocity and flow angel at the diffuser inlet; r, C, α is the current values of the radius, velocity and flow angel.

Moment of forces acting on the vanes zone from r_{in} to r with width b is determined by the formula

$$\Delta M = z_{\mathrm{v}} \cdot b \cdot \int_{r_{in}}^{r} \Delta p \cdot r \cdot dr, \qquad \text{Eq. (2)}$$

where $\Delta p = \left(p_{ps} - p_{ss} \right)$ is the pressure difference on the pressure and suction surfaces of the vane, z_{v} is the quantity of vanes.

After the transformations of Eq. (2) using gas-dynamic functions for isentropic flow we get

$$r_{in} \cdot \lambda_{in} \cdot \cos\alpha_{in} - r \cdot \lambda \cdot \cos\alpha = \frac{b \cdot z_{\mathrm{v}} \cdot p_{in}^{*} \cdot \sigma}{\dot{m} \cdot a_{kr}} \int_{r_{in}}^{r} \left[\pi\left(\lambda_{ps}\right) - \pi\left(\lambda_{ss}\right) \right] \cdot r \cdot dr, \qquad \text{Eq. (3)}$$

where a_{kr} is the critical speed of sound, $\lambda = C / a_{kr}$ is the reduced velocity (coefficient of velocity), p_{in}^{*} is the pressure of adiabatically stagnated flow at the diffuser inlet, $\sigma = 1 - (1 - \sigma_{out}) \cdot \dfrac{r - r_{in}}{r_{out} - r_{in}}$ is the total pressure loss factor, $\sigma_{out} = p_{out}^{*} / p_{in}^{*}$, $\pi\left(\lambda\right) = \left(1 - \dfrac{k-1}{k+1} \cdot \lambda^{2} \right)^{\frac{k}{k-1}}$ is gas-dynamic pressure function, k is the adiabatic index.

The flow equation for the annular diffuser element width b is as follows

$$\dot{m} = C_{r} \cdot \rho \cdot 2\pi \cdot r \cdot b \cdot \tau, \qquad \text{Eq. (4)}$$

where τ – is the blockage factor.

The blockage factor is determined by the formula

$$\tau = 1 - \frac{\delta' \cdot z_{\mathrm{v}}}{2\pi \cdot r \cdot \sin\alpha}, \qquad \text{Eq. (5)}$$

where $\delta' = \delta_{\mathrm{v}} + \Sigma\delta^*$ is the given blade thickness; $\Sigma\delta^*$ is the total displacement thickness of the boundary layers in the interblade channel.

The flow equation using gas-dynamic functions is:

$$\dot{m} = \lambda \cdot \varepsilon(\lambda) \cdot a_{kr} \cdot \rho^* \cdot 2\pi \cdot r \cdot b \cdot \tau \cdot \sin\alpha, \qquad \text{Eq. (6)}$$

where $\varepsilon(\lambda) = \left(1 - \frac{k-1}{k+1} \cdot \lambda^2\right)^{\frac{1}{k-1}}$ is the gas-dynamic density function; ρ^* is the density of the adiabatically stagnated flow.

Eq. (3) and Eq. (6) form a system of equations with unknowns $\alpha, \lambda, \lambda_{ps}, \lambda_{ss}$. The reduced velocities on the pressure (λ_{ps}) and suction (λ_{ss}) surfaces of the vane and the average velocity are interconnected. For example, for a linear law, the pressure changes along the grid step are $\pi(\lambda_{ps}) + \pi(\lambda_{ss}) = 2 \cdot \pi(\lambda)$.

With a given velocity value on the pressure surfaces of the vane, velocity value on the suction surfaces of the vane is determined by the ratio

$$\pi(\lambda_{ss}) = 2 \cdot \pi(\lambda) - \pi(\lambda_{ps}). \qquad \text{Eq. (7)}$$

Thus, a system of two equations has two unknown values – average velocity λ and the flow angle α , that can be determined by any numerical method.

An important part of the vane diffuser design method is to determine the optimum velocity distribution. The velocity distribution is determined by the condition that the turbulent boundary layer is predetermined.

Determination of velocity distribution

The estimation of the possibility of appearance of a viscous flow separation can be carried out by integrating the angular momentum equation for the boundary layer (Karman equation), which has the form:

$$\frac{d\theta}{dl} + \frac{1}{C} \cdot \frac{dC}{dl} \cdot \theta \cdot (2 + H) + \frac{1}{\rho}\frac{d\rho}{dl} \cdot \theta = \frac{\tau_w}{\rho C^2}, \qquad \text{Eq. (8)}$$

where $H = \delta^*/\theta$ is the formparameter; C, ρ is velocity and density at the boundary of the boundary layer at $y = \delta$ To determine the separation, the criteria for separation are as follows:

$$f = \frac{dC}{dl} \cdot \frac{\theta}{C} G\left(\mathrm{Re}^{**}\right). \qquad \text{Eq. (9)}$$

The condition of separation is $f \le f_{kr}$. In this paper, the Loitcyanskii method was adopted according to which $G\left(\mathrm{Re}^{**}\right) = \left(\mathrm{Re}^{**}\right)^{1/6}$ and $f_{kr} = -0{,}02$, $\mathrm{Re}^{**} = C \cdot \theta/\nu$.

The relative momentum thickness of boundary layer is determined from the ratio:

$$\bar{\theta} = 0,0159 \cdot \mathrm{Re}^{-0,15} \cdot \lambda^{-3,55} \cdot \left(\int_0^{\bar{l}} \lambda^4 \cdot d\bar{l} \right)^{0,85}, \qquad \text{Eq. (10)}$$

where $\mathrm{Re} = C \cdot l / \nu$ – Reynolds number.

The formula for determining the velocity distribution that provides the preseparation state of the turbulent boundary layer is determined by integrating the momentum equation for the boundary layer and has the form:

$$\lambda = \lambda_1 \cdot \left[1 + \frac{\left(\bar{l} - \bar{l}_1\right) \cdot \left(2 + H_s\right) \cdot \left(-f_{kr}\right)}{\bar{\theta}_1} \right]^{-\frac{1}{2+H_s}}, \qquad \text{Eq. (11)}$$

where λ_1 and $\bar{\theta}_1$ is the reduced velocity (coefficient of velocity) and the relative momentum thickness at the distance $\bar{l}_1 = l_1 / L = 0,02...0,05$ from the inlet edge of the vane, H_s is the formparameter.

Designing the vane diffuser

The solution of Eq. (3) and Eq. (6) is performed by the method of successive approximations. The integral in Eq. (3) is solved by any numerical method, for example, the method of central rectangles.
Eq. (11) is used to determine the velocities at the pressure surfaces of the vane. Then, using Eq. (7) with Eq. (3) the unknown λ_{ss} is eliminated.

The result of a joint solution of Eq. (3) and Eq. (6) for a number of sections from the inlet to the outlet of the diffuser is the distribution of the rate-average flow angles and velocities along the vane.

Based on this data, the geometric parameters of the vane and the velocity distributions on its surfaces are determined. Boundary layers are taken into account in the calculations. The area of the boundary layer is replaced by the displacement thickness, which is determined by the ratio $\delta^* = H_s \cdot \delta^{**}$. The formula Eq. (10) determines the values of the relative momentum thickness and then the formula Eq. (9) determines the value of the formparameters on the suction surfaces of the vane. Often, the suction surfaces of the vanes have a flow separation, but the separation occurs much further from the inlet edges of the vanes than in the diffusers with vanes with the traditional shape of the profile. Obviously, head losses will be smaller.

Designing the channel diffuser

On the pressure surfaces of the segments, the law of change of velocity is given for the predetermined state of the turbulent boundary layer according to formula Eq. (11).

The law of change of pressure in the circular direction in the channel and the linear law of change of the average angle of flow along the channels are used $\alpha = f(r)$. Then, as a result of solving the system Eq. (3) and Eq. (6) the change in the average flow velocity and the blockage factor along the radius of the diffuser are determined. The calculation is performed using numerical methods.

The calculation of the momentum thickness in the boundary layer on the surfaces of the segments is performed by the formula Eq. (10), and the value of the formparameter is calculated by the formula Eq. (9). After calculating the boundary layer displacement thicknesses in the channel, the thickness of the diffuser segments is determined by Eq. (5). The end result of the design calculation is to get dependencies $\delta_s = f(r)$. Assuming the angle of the midline of the segments is equal to the average angle of the flow ($\alpha_s = \alpha$), we have a completely specified shape of the segments of the channel diffuser.

NUMERICAL CALCULATION

Simulation of gas flow in the flow part of the centrifugal compressor stage and determination of diffuser efficiency was performed in the ANSYS CFX software package for Reynolds numbers Re $>10^5$ and Mach numbers 0,2…0,6. The calculations were performed for 8 modes of operation of stages, which differed by gas consumption.

Fig. 1 shows the vane profiles.

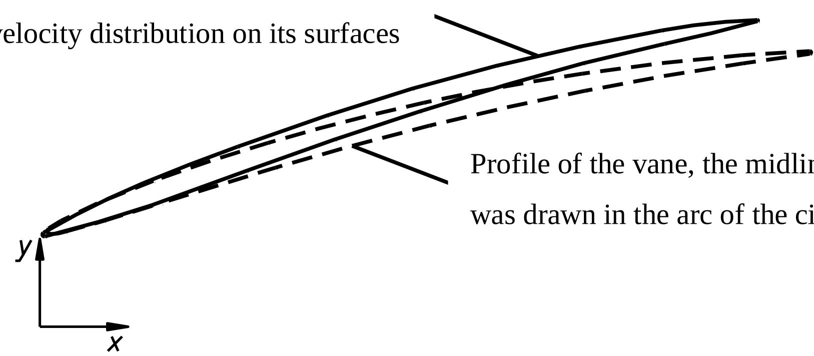


Figure 1: Profiles of vanes

Fig. 2 shows the values of total pressure loss and static pressure rise coefficients for diffusers designed by different methods.

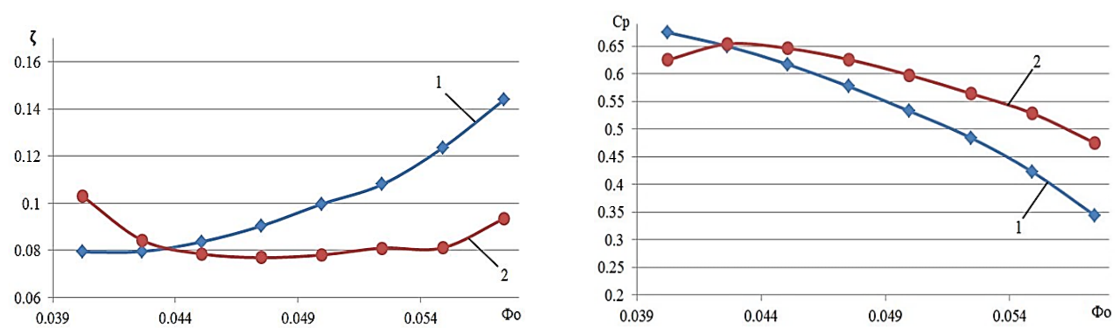
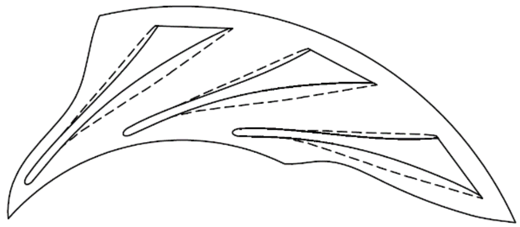


Figure 2: Dependences of loss coefficients and static pressure rise coefficients on flow coefficient; 1–the middle line of the vane is an arc of a circle; 2–given velocity distribution on the surface of the vane

Vane diffuser, designed with a predetermined velocities distribution, has better performance in nominal mode ($\Phi_0 = 0{,}0502$) up to 28%, as well as in all modes with flow rate higher than nominal up to 54%. The static pressure rise coefficient of the vane diffuser, designed using the proposed method, has a higher value at nominal mode of 9% and all modes with a flow rate of more than 21%. The prospect of the proposed vane diffuser profiling method with a given velocity distribution on the vanes surfaces is obvious.

In the ANSYS CFX software, a channel diffuser was calculated using the proposed method. The characteristics of the traditional channel diffuser and channel diffuser, which profile was obtained using the above method, are compared. The diffusers have the same geometric dimensions ($r_3 = 253$ mm, $r_4 = 358$ mm, $b_3 = b_4 = 20$ mm, the thickness of the segments at the inlet $\delta_{s\,in} = 7$ mm, $z_s = 14$, $\alpha_{in} = 14^0$, $\alpha_{out} = 44^0$), but different segment profiles. Fig. 3 shows segment profiles and diffuser characteristics obtained in ANSYS CFX.



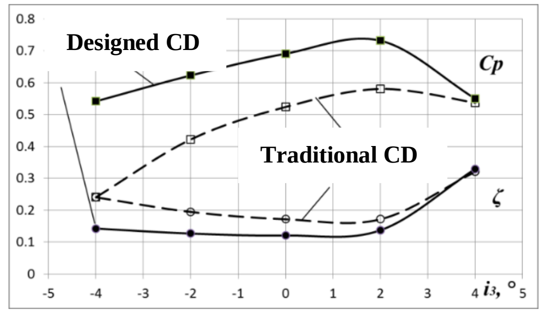


Figure 3: Profiles and characteristics of traditional geometry CD (dashed line) and designed using the above method (solid line)

Designed channel diffuser is more efficient than traditional, practically in all modes. In the calculated mode ($i_3 = \alpha_{3s} - \alpha_3 = 0^0$) for this diffuser, the value of the static pressure rise coefficient $C_p = \left(p_{out} - p_{in}\right)/\left(p_{in}^* - p_{in}\right)$ is higher by 32% and the total pressure loss coefficient $\zeta = \left(p_{in}^* - p_{out}^*\right)/\left(p_{in}^* - p_{in}\right)$ is lower by 30%. At negative angles of attack ($i_3 < 0^0$) the losses in the designed CD are reduced by almost 2 times in comparison with the traditional one; at positive angles of attack, the difference of the values of the coefficients C_p and ζ is less significant for these diffusers.

EXPERIMENTAL RESEARCH

Experimental investigations of the diffuser model, which was designed using presented method, were made at the aerodynamic stand. Fig. 4 shows scheme of experimental model.

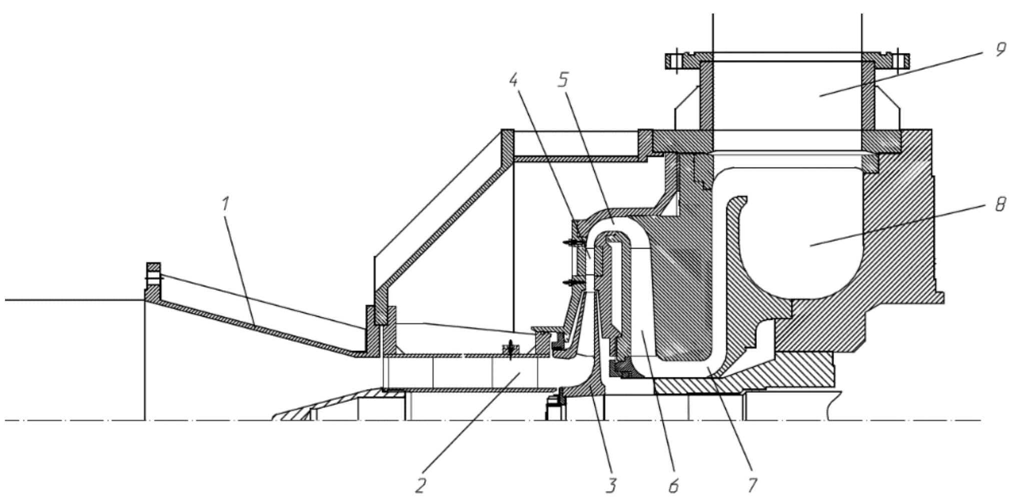


Figure 4: Scheme of the experimental model

The design pressure distributions along the midline and along the surfaces of the CD segments were confirmed experimentally. Fig. 5 shows the distribution of relative static pressures along the surfaces of segments.

The discrepancy between theoretical and experimental values of the pressures at the corresponding points does not exceed 2%.

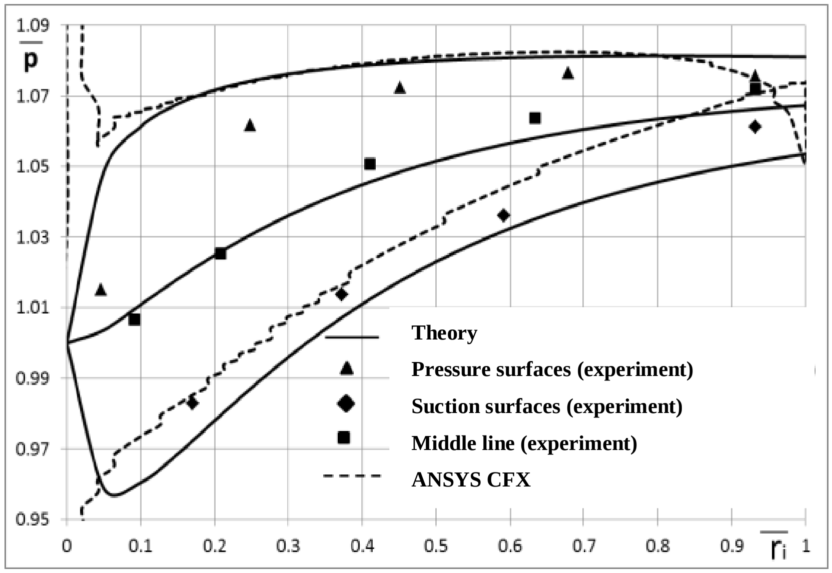
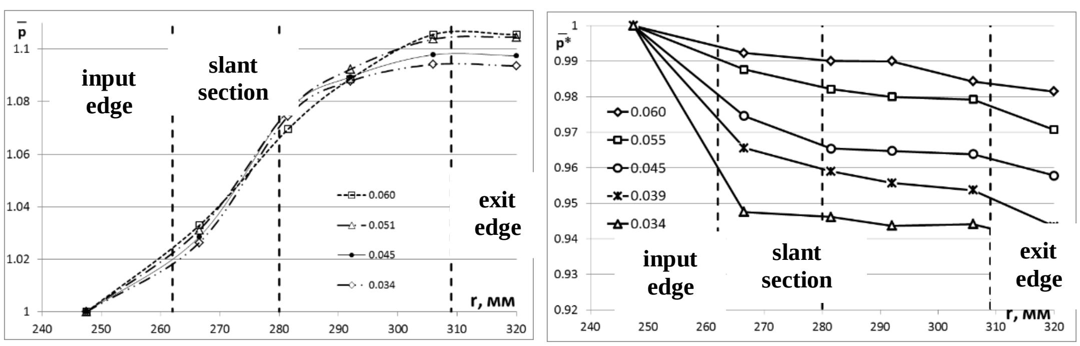


Figure 5: The distribution of relative static pressures along the surfaces of segments

Therefore, the method presented in this paper allows channel diffuser segments to be profiled with sufficient accuracy for engineering calculations. The calculated pressure distributions can be used to qualitatively and quantitatively evaluate the flow structure in a diffuser, which is important at the design stage.



a) b)

Figure 6: The distributions of relative static pressures (a) and relative total pressures (b) on the middle line along the radius CD at different values of the flow rate Φ_0.

Fig. 6 shows the distributions of relative static pressures and total pressures at different values of the flow rate Φ_0 . The relative static pressure is defined as the ratio of the static pressure to the static pressure at the inlet to the channel $\overline{p} = p/p_{in}$.

The pressure is increasing most intensively in the slant section. The values of the pressure at the vaneless section in front of the inlet edges of the segments and in the slant section are independent from the flow. The effect of the flow rate on the pressure becomes noticeable from the middle of the diffuser`s channel. As the flow rate increases, the pressure increases. This may be due to the fact that the flow at the outlet of the impeller in the case of high flow rates is more even in the grid step. Therefore, the conversion of velocity to pressure is more efficient with a low level of energy loss associated with flow levelling. The pressure at the exit edges is hardly at all increasing.

A significant drop in the total pressure (up to 5%) occurs on the bladeless section before the diffuser.

The lower the mass flow rate is, the greater is the pressure drop. This is mainly due to the mixing of "jets" and "traces" formed on the impeller.

The total pressure drop in the diffuser channels is close to 1% for almost all compressor modes, which indicates the high efficiency of the segment profiles. Losses in the CD are also increased due to the mixing of "traces" along the exit edges of the segments.

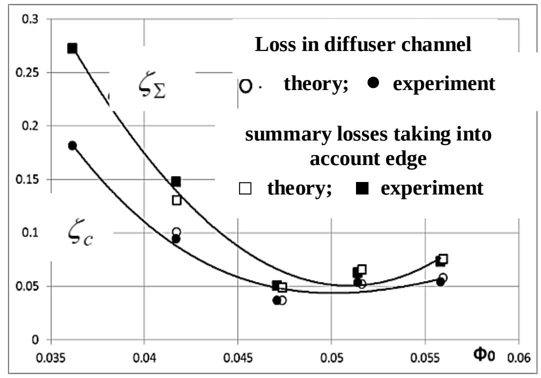


Figure 7: Dependences of loss coefficients CD on flow coefficient

Figure 7 shows experimental and theoretical loss coefficients obtained for different sections of the CD. The discrepancy between the experimental and theoretical values of the loss coefficients ζ_c, ζ_Σ does not exceed 15% for all modes of operation. The lowest CD loss is close to the nominal mode with $\Phi_{0n} \approx 0{,}047$. With the reduction of the mass flow of gas $\Phi_0 < \Phi_{0n}$ the losses increase more intensively than in the case of an increase in flow rate.

CONCLUSIONS

A new method of designing vane diffusers with a predetermined velocity distribution on vane surfaces provides an improvement in the gas-dynamic characteristics of diffusers compared to traditional geometry diffusers, which is reduced losses from 28% to 50% and increased values of static pressure rise coefficients from 9% to 21% in a wide range of modes. The proposed method for profiling centrifugal compressor channel diffusers allows to reduce the calculated mode losses by 30% and to increase the value of the static pressure rise coefficient by 32% compared to traditional geometry channel diffusers.

NOMENCLATURE

$\dot{m}$	mass flow (kg×s–1)	λ	reduced velocity
r_{in}	radius at the diffuser inlet (m)	τ	blockage factor
C_{in}	velocity at the diffuser inlet (m)	δ^*	displacement thickness (m)
α_{in}	flow angel at the diffuser inlet (grad)	θ	momentum thickness (m)
r	current values of the radius (m)	f	formparameter
C	current values of the velocity (m×s–1)	H	formparameter
α	current values of the flow angel (grad)	i_3	angle of attack (grad)
p_{ps}	pressure on the pressure surfaces of the vane (Pa)	C_p	static pressure rise coefficient
p_{ss}	pressure on the suction surfaces of the vane (Pa)	ζ	total pressure loss coefficient
b	width of the diffuser (m)	Φ_0	flow coefficient

REFERENCES

Stratford, B S., 1959. An experimental flow with zero skin friction throughout its region of pressure rise. ASME J. Fluid Mech. 5, 17-35.

Kalinkevych, M., Skoryk, A., 2013. Flow and performance investigation of the specially designed channel diffuser of centrifugal Compressor. 8th International Conference on Compressors and their Systems. City University London, UK, 489-500.